Impact of Unsteady Secondary Air Flow Interaction with Main Flow on Loss Generation in Axial Turbines

by

David Clifton

B.S.E., Princeton University (2012)

Submitted to the Department of Aeronautics and Astronautics in partial fulfillment of the requirements for the degree of Master of Science in Aeronautics and Astronautics at the MASSACHUSETTS INSTITUTE OF TECHNOLOGY September 2014

© Massachusetts Institute of Technology 2014. All rights reserved.

Signature redacted

Author .................................................................
Department of Aeronautics and Astronautics
August 21, 2014

Signature redacted

Certified by .............................................................
Choon S. Tan
Senior Research Engineer of Aeronautics and Astronautics
Thesis Supervisor

Signature redacted

Accepted by .........................................................
Paul C. Lozano
Associate Professor of Aeronautics and Astronautics
Chair, Graduate Program Committee
Impact of Unsteady Secondary Air Flow Interaction with Main Flow on Loss Generation in Axial Turbines

by

David Clifton

Submitted to the Department of Aeronautics and Astronautics on August 21, 2014, in partial fulfillment of the requirements for the degree of Master of Science in Aeronautics and Astronautics

Abstract

Secondary air, often called purge air, is injected through the endwall gap between stationary vanes and rotating rotors in axial turbines to prevent ingestion of the hot working gas into the endwall cavities. Three-dimensional flow computations, unsteady as well as steady, have been carried out to assess the role of vane-rotor stage induced flow unsteadiness on loss generation from purge flow interacting with main flow. Computational models of varying physical and geometry complexity that range from simple bladeless annulus configurations to vane-rotor stage configurations were used for identifying the specific flow features responsible for loss generation; these were also complemented by control volume analyses of simple two streams flow model. Computed results showed that the vane induced circumferentially non-uniform static pressure field, and to a much lesser extent rotor blade induced bow waves, significantly increases loss generation upon introduction of injected flow with and without swirl; any flow approximations that renders the pressure field imposed on the purge flow as circumferentially uniform would underestimate the loss due to purge flow-main flow interaction. Flow unsteadiness induced by vane-rotor interaction has marginal, if any, impact on loss generation from introducing purge flow into the main flow path. Judicious flow path modification such as endwall contouring that reduces or eliminates vane-induced static pressure circumferential non-uniformity imposed upon the purge flow has been shown to significantly reduce the purge flow driven loss. Computed results also show that loss generation increases linearly with injected mass flow rate (7.5% more loss per 1% purge flow) and decreases quadratically with purge flow swirl until it matches the main flow circumferential velocity (85% reduction in purge flow induced based loss at that point). Purge flow induced loss has a weak dependence on the width of the purge duct and angle of injection.

Thesis Supervisor: Choon S. Tan
Title: Senior Research Engineer of Aeronautics and Astronautics
Acknowledgments

It goes without saying that the work presented in this thesis would not have been possible without the help and support of a number of other people. I would like to acknowledge their contributions and express my gratitude for their help over the past two years.

First I will thank my research adviser, Dr. Choon Tan. He kept me on course and served as my go to source whenever I encountered a problem I could not solve on my own. On the rare occasion he did not know the definitive answer he always had the humility to recognize it and the wisdom to point me in the right direction. Even when things were difficult his office door was always open. For these qualities and the help he has given me over these past two years, I am grateful. Additionally, I wish to express my graditude to the other member of the Gas Turbine Laboratory who has been immensely helpful during the course of my time at MIT, Timothy Palmer. He shared his thoughts, his suggestions, and his time freely and in doing so saved me much grief. He was a font of knowledge on almost every topic I encountered, and his experience was invaluable.

I am also grateful to the members of Siemens Energy who helped me over the course of these past two years. First, I must thank Dr. Matthew Montgomery for taking the time to share his wisdom and experience on all matters. Next, Anthony Malandra, who was always patient and asked questions that helped guide my thinking. I am also indebted to Humberto Zuniga for his aid and guidance when meshing and convergence difficulties threatened to stymie my progress. Without his support I would have fallen well short of my goals. Lastly, I must thank Siemens Energy for funding this work, as without this sponsorship none of what I’ve accomplished over the past two years would have been possible.

Finally, I thank my family, as without their support I never could have made it to this day. My father, for his calm temperment and even-keeled advice. My mother, for her common sense advice and willingness to lend an ear. And my sister, for reminding me of the world outside.
## Contents

1 Introduction
   1.1 Literature Review ........................................ 19
   1.2 Technical Goal ........................................... 21
      1.2.1 Specific Technical Objective and Research Questions 21
   1.3 Contributions and Findings .................................. 22
   1.4 Organization ........................................... 23

2 Research Approach
   2.1 Phenomena-Targeted Modeling ................................ 25
      2.1.1 3D Bladed Full Sector .................................. 26
      2.1.2 Two-Dimensional Axisymmetric Model .................... 32
      2.1.3 3D Bladeless Annular Model ............................. 34
   2.2 CFD Simulation Solver Specifications and Guidelines ....... 39
      2.2.1 Steady State Calculation ............................... 39
      2.2.2 Unsteady Simulation ................................. 40
   2.3 Summary ........................................... 40

3 Entropy Based Loss Accounting ................................ 43
   3.1 Introduction ........................................... 43
   3.2 Entropy Based Loss Accounting ................................ 43
   3.3 Viscous Lost Work ........................................ 45
      3.3.1 Viscous Lost Work in Non-Uniform Streams .......... 46
      3.3.2 Current Implementation of Viscous Lost Work Formulation 48

7
3.4 Time Averaging of Unsteady Flow
3.4.1 Evaluating Time-Average Loss
3.5 Attributing Loss to Flow Phenomena
3.5.1 Geometric Bounding Method
3.5.2 Two Stream Mixing Method
3.6 Summary

4 Unsteady Flow and its Time Averaged Impact
4.1 Net Loss Trends
4.1.1 Viscous Lost Work Profiles
4.1.2 Removal of Purge Slot
4.2 Loss Breakdown by source
4.2.1 Viscous Shear Layer
4.2.2 Bladeless Annular Model
4.2.3 Full Sector - Secondary Flow Region
4.2.4 Other Loss Sources
4.3 Summary

5 Mechanism for Increased Loss Generation in the Shear Layer Due to Purge-
Main Flow Interaction
5.1 Unsteady NGV-Rotor Interaction
5.1.1 2D Axisymmetric Model
5.1.2 3D Bladeless Annular Model
5.2 Circumferential Variations in Purge Flow
5.2.1 3D Bladeless Annular Model
5.2.2 Frozen Rotor Model
5.3 Summary

6 Two Stream Mixing and Parametric Analysis
6.1 Mixing of Two Streams
6.1.1 Two Stream Mixing in the Bladeless Annular Model
6.1.2 Full Sector Mixing Model ........................................... 89
6.2 Parametric Analysis ................................................. 90
  6.2.1 Purge Mass Flow Rate ........................................... 92
  6.2.2 Purge Flow Swirl ............................................... 94
  6.2.3 Purge Slot Width and Injection Angle ...................... 95
6.3 Summary ................................................................. 98

7 Strategies to Reduce Purge Flow Circumferential Variation 101
  7.1 Bladeless Annular Model Endwall Contouring ................. 102
    7.1.1 Characteristics of NGV-Induced Circumferentially Varying Pressure 102
    7.1.2 Rotor Endwall Axisymmetric Modification Description and Sizing . 103
    7.1.3 Results from Modified Rotor Endwall ....................... 106
    7.1.4 Axisymmetric Bladeless Annular Stator Endwall Modification . 107
    7.1.5 Results from Modified Stator Endwall Bladeless Annular Model . 109
  7.2 Suggestions and Recommendations for Full Sector Endwall Shaping . 112
    7.2.1 Full Sector Rotor Endwall Contouring .................... 112
    7.2.2 Full Sector Stator Non-Axisymmetric Endwall Contouring .... 112
  7.3 Summary .............................................................. 115

8 Conclusion and Future Work 117
  8.1 Summary .............................................................. 117
    8.1.1 Key Findings .................................................. 118
  8.2 Future Work ......................................................... 119
# List of Figures

1-1 **Diagram of main and purge flows in the first stage of an axial turbine** 20

2-1 Full sector computational mesh ................................. 28

2-2 Single rotor passage computational mesh .......................... 29

2-3 Diagram of purge duct in hypothetical turbine ...................... 29

2-4 Purge duct schematic highlighting geometric parameters .......... 30

2-5 Circumferential view of rotor endwall modification to bladeless annular mesh 32

2-6 Depiction of asymmetric stator endwall contouring ...................... 33

2-7 Circumferential view of stator endwall modification to bladeless annular mesh ............................... 33

2-8 Original 2D mesh designed by Zlatinov .............................. 34

2-9 Revised 2D mesh .............................................. 35

2-10 3D view of bladeless annular mesh ................................ 37

2-11 Bladeless annular mesh overlaid with 3D full sector ...................... 38

3-1 T-s diagram of multi-stream expansion illustrating the concept of work-averaged pressure ........................................ 46

3-2 Schematic showing main flow, purge flow, and mixed flow .............. 52

3-3 Two Stream Mixing Measurement Regions for Bladeless Annular Case ..................... 53

3-4 Two Stream Mixing Measurement Regions for Full Sector Case ..................... 53

4-1 Full sector model: cumulative viscous lost work profile, varying mass ............................. 58

4-2 Full sector model: cumulative viscous lost work profile, varying swirl ............................. 59
4-3 Full sector model - effect of purge slot with zero mass flow rate on loss generation ........................................ 60
4-4 Full sector model - cumulative viscous lost work profile, 0% purge mass .... 61
4-5 Definition of regions each with specific loss generating flow features .... 62
4-6 Near hub pressure plot showing regions of purge flow being blocked for 2% purge mass flow ....................................... 65
4-7 Purge Flow Velocity Profile .............................................. 66
4-8 Full sector asymmetry visualization via purge flow tracking - 2% purge mass flow .................................................. 67
4-9 Near hub pressure plot showing regions of purge flow being blocked for 0.5% purge mass flow ........................................ 68
4-10 Bladeless annular model - cumulative viscous lost work under varying parameters ............................................. 70
4-11 Separation region - steady flow ............................................. 72
4-12 Separation region - unsteady flow ........................................... 72
4-13 Steady tip clearance loss comparison .......................................... 73
4-14 Unsteady tip clearance loss comparison ......................................... 74
5-1 2D axisymmetric flow model: steady vs. unsteady cumulative viscous lost work for varying mass ..................................................... 77
5-2 2D axisymmetric flow model: steady vs. unsteady cumulative viscous lost work for varying swirl ................................................. 78
5-3 Bladeless annular flow model: steady vs. unsteady cumulative viscous lost work ................................................................. 79
5-4 Bladeless annular steady comparison of loss - symmetric vs. asymmetric ...... 81
5-5 Full sector comparison of loss in unsteady simulation vs. frozen rotor simulation ............................................................... 82
6-1 Schematic showing main flow, purge flow, and mixed flow ...................... 87
6-2 Mixing Code Measurement Regions for Bladeless Annular Case ............. 88
6-3 Mixing Code Measurement Regions for Full Sector Case .................... 89
6-4 Full sector, purge and main flow velocity profiles under steady and unsteady simulation ......................................................... 90
6-5 Full sector, profile for difference in purge flow and main flow velocity components from steady and unsteady computations. .................. 91
6-6 Bladeless annular model, purge and main flow velocity profiles under steady and unsteady simulation ................................. 92
6-7 Bladeless annular model, profile for difference in purge flow and main flow velocity components from steady and unsteady computations. 93
6-8 Purge duct schematic highlighting geometric parameters .......................................................... 94
6-9 Parametric loss analysis on purge flow mass .......................................................... 95
6-10 Parametric loss analysis on purge flow swirl .......................................................... 96
6-11 Purge duct schematic of modified duct width .......................................................... 98
6-12 Purge duct schematic of modified duct angle .......................................................... 98

7-1 NGV-induced pressure field exponential regression .......................................................... 104
7-2 Depiction of stator pressure influence on near hub and purge flow .......................................................... 105
7-3 Circumferential view of rotor endwall modification to bladeless annular mesh .................. 106
7-4 Rotor endwall modification - comparison of circumferential profiles of purge velocity .................................................................. 107
7-5 Circumferential view of stator endwall modification to bladeless annular mesh .................. 108
7-6 Stator endwall modification - comparison of circumferential profiles of purge velocity .................................................................. 110
7-7 Modified Stator Endwall - Contours of loss generation, vectors of velocity.
Shows separation region. ................................................................ 111
7-8 Schematic drawing of full sector proposed stator endwall contouring .................. 113
7-9 Stator endwall schematic emphasizing relevant sizing parameters ................. 114
### List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Characteristics of a Representative Turbine Stage</td>
<td>27</td>
</tr>
<tr>
<td>2.2</td>
<td>Table of Computational Models and Flow Configurations Used in CFD, Uniform Denotes Circumferentially Uniform While Varying Denotes Circumferentially Varying</td>
<td>41</td>
</tr>
<tr>
<td>4.1</td>
<td>Full Sector Viscous Lost Work</td>
<td>57</td>
</tr>
<tr>
<td>4.2</td>
<td>Full Sector Loss Source Attribution</td>
<td>63</td>
</tr>
<tr>
<td>4.3</td>
<td>Viscous Lost Work in Bladeless Annular Model</td>
<td>69</td>
</tr>
<tr>
<td>6.1</td>
<td>Bladeless Annular Model: CFD vs. CV</td>
<td>91</td>
</tr>
<tr>
<td>6.2</td>
<td>Full Sector Model: CFD vs. CV</td>
<td>93</td>
</tr>
<tr>
<td>6.3</td>
<td>Full Sector Parametric Analysis - Purge Mass Flow Rate</td>
<td>94</td>
</tr>
<tr>
<td>6.4</td>
<td>Parametric Trend in Loss with Purge Swirl from Full Sector Steady and Unsteady Computations</td>
<td>97</td>
</tr>
<tr>
<td>6.5</td>
<td>Variation in Loss with Purge Slot Width and Purge Flow Injection Angle Computed Using Bladeless Annular Model</td>
<td>97</td>
</tr>
</tbody>
</table>
Nomenclature

List of Symbols

\( B.A. \)  Bladeless Annular
\( c_p \)  Specific heat at constant pressure
\( \dot{e} \)  Shear strain rate
\( h_t \)  Specific total enthalpy
\( k_{eff} \)  Effective thermal conductivity
\( \dot{m}_{avg} \)  Mass flow rate radially or axially averaged on a surface to a point
\( \dot{m}_{total} \)  Total mass flow rate through a surface
\( M \)  Mach Number
\( NGV \)  Nozzle Guide Vane
\( P_t \)  Total pressure
\( P_t^{wa} \)  Work averaged total pressure
\( \dot{S}_{gen}^{'''} \)  Volumetric entropy generation rate
\( \dot{S}_{visc}^{'''} \)  Volumetric entropy generation rate due to viscous sources
\( \dot{S}_{therm}^{'''} \)  Volumetric entropy generation rate due to thermal sources
\( T \)  Temperature
\( TA \)  Denotes time averaged quantity
\( T_t \)  Total temperature
\( T_t^{ma} \)  Mass averaged total temperature
\( \vec{u} \)  Velocity vector
\( U \) Disk rim speed
\( v_x \) Velocity - axial component
\( v_r \) Velocity - radial component
\( v_\theta \) Velocity - circumferential component
\( w \) Purge blockage circumferential length

\( W_{\text{ideal}} \) Ideal work output
\( W_{\text{visc}} \) Viscous lost work
\( \epsilon \) Eddy viscosity
\( \lambda \) Stator pitch
\( \mu \) Dynamic viscosity
\( \Omega \) Angular speed of rotating component

**Non-Dimensional Quantities**

\( \gamma \) Ratio of specific heats
\( \eta \) Efficiency
\( \Lambda \) degree of reaction, \( \frac{\Delta h_{\text{rotar}}}{\Delta h_{\text{stage}}} \)

\( \Phi \) Flow Coefficient, \( \frac{v_x}{\Omega_{\text{midspan}}} \)
\( \pi \) Stage pressure ratio
\( \Psi \) Stage Loading, \( \frac{\Delta h}{(\Omega_{\text{midspan}})^{\frac{\gamma}{\gamma-1}}} \)
\( \sigma \) Solidity, (chord length/pitch)
Chapter 1

Introduction

Gas turbines for power generation have become increasingly prevalent due to their relative flexibility. Turbine inlet temperatures have been steadily increased with the goal of higher cycle efficiency, and consequently the working fluid is well above the melting temperature of the structural material. Coatings are used to protect the structure, but the temperature can be raised even higher than they would allow through blade cooling using air bled from the final compressor stages. Another concern is the gaps interspacing the rotating and stationary vanes - ingestion of hot gasses out of the main flow path and into the inner wheelspace cavity could lead to significant to drastic shortening of component life or even failure. An example machine is shown in figure 1-1. To avert this, compressor air, dubbed "purge flow" is ejected from said gaps. However, this is another source of lost work, and investigation into the origins of the loss may provide means by which it can be reduced.

1.1 Literature Review

The current literature on purge flow interaction with main flow was helpful in guiding the flow of the research effort yielding the results presented in this thesis. The single dominant source from which this thesis drew was the the research presented in the Thesis and related paper by Metodi Zlatinov, the predecessor to the current efforts in this thesis [1] [2]. A brief synopsis of the most relevant parts of his literature review follows. The significant majority of studies show a decrease in efficiency associated with purge flow injection, though Pau
et. al. [3] found that an increase in efficiency was possible. However, that increase was attributed largely to modifications to a downstream shock system, a feature not present in the turbine under consideration here. Ong et. al. [4] applied a control volume analysis to estimate the losses due to main flow - purge flow mixing. Lastly, Zlatinov listed a number of publications [5] [6] [7] that noted a decrease in mixing loss between main and purge flows when the purge flow is swirled prior to injection.

Extending from Zlatinov’s review, Jenny et. al [8] investigated unsteady interaction of passage vortices through both experimental and computational methods, in addition to earlier work specifically relating to purge flow interaction with main flow [9]. Pullan found that unsteady rotor passage features, including vortices, are stationary with respect to the stator and thus attributable to the vane. Furthermore, he found that unsteady NGV-rotor interaction can increase loss generation [10]. Langston et. al. provided experimental data examining crossflows and vortex movement and interaction with hub and shroud endwall boundary layers [11] [12].
Clearly, a substantial amount of research has been done on secondary air losses. Furthermore, Zlatinov established a relationship between introduction of purge flow and additional loss generation for steady flow. However, the impact of flow unsteadiness on loss generation associated with flow interaction of main and purge flows remains to be addressed and assessed quantitatively.

1.2 Technical Goal

The research described in this thesis constitutes a direct follow-on to the work of Zlatinov [1] [2]. Zlatinov focused on the loss generation associated with the introduction of purge flow into the main flow path of a turbine. In his analysis, the flow is rendered steady through the use of a mixing plane located at an axial location between the vane trailing edge and the leading edge of the purge slot. However, the flow in the turbine main flow path is inherently unsteady due to the relative motion between the vanes (NGV) and the blades (rotor). As such there is a need to assess the role of flow unsteadiness on loss generation associated with the introduction of purge flow. And furthermore, there is also a need to determine to what extent the findings by Zlatinov hold up in an unsteady flow environment representative of an NGV-rotor stage.

1.2.1 Specific Technical Objective and Research Questions

The specific technical objective is to assess the role of flow unsteadiness induced by NGV-rotor interactions on loss generation associated with the introduction of purge flow into the turbine main flow path. To accomplish this stated goal, the following research questions need to be answered:

1. How do the loss sources identified by Zlatinov [1] change in behavior or magnitude when unsteady effects are included?

2. Does unsteadiness introduce any entirely new effects? If so, what are the quantitative impacts on loss?
3. Based upon the new understanding of purge flow dependent loss sources under unsteady conditions; what new design guidelines or scaling laws can be formulated?

1.3 Contributions and Findings

The key findings are:

1. NGV induced circumferentially non-uniform static pressure field (and to a much lesser extent rotor blade induced bow waves) significantly increases loss generation in the turbine flow path upon introduction of injected purge flow with and without swirl. A mixing plane upstream of the purge slot, which renders the pressure field imposed on the purge slot circumferentially uniform, leads to an underestimation of the loss associated with purge flow - main flow interaction.

2. Flow path modification (such as endwall contouring) that reduces or eliminates NGV-induced static pressure circumferential non-uniformity imposed upon the purge flow was shown to reduce circumferentially varying purge flow driven loss by $\sim 75\%$ or more. Example: if baseline loss (computed using mixing plane approximation) is 1.0, and loss computed with mixing plane removed is 1.08, flow path modifications can reduce it to 1.02.

3. Flow unsteadiness introduced by NGV-rotor interaction has marginal, if any, impact on loss generation from the introduction of purge flow into the main flow path. This is reflected in the frozen rotor simulation, which yield a nearly identical loss as that from the unsteady simulation; the frozen rotor computations encompasses circumferentially varying purge flow driven loss but excludes unsteady flow driven loss.

4. Loss increases linearly with additional purge mass flow under both steady and unsteady conditions. Steady state analysis showed that a 1% additional purge mass flow leads to approximately 8.5% additional loss. In unsteady flow analysis 1% additional purge mass flow leads to approximately 7.5% additional loss. The difference in loss per mass flow occurs because the circumferential non-uniformity (primary new phenomena between the two situations) is sensitive to the quantity of injected mass flow
and has proportionally greater effect with smaller purge mass flow rate. With purge flow circumferential velocity matching the main flow, the loss associated with that purge flow can be decreased by up to 85% relative to the situation of purge flow with no swirl. The purge flow induced loss has a quadratic dependence on the purge flow swirl (As the main flow swirl remains constant as the purge flow swirl is varied).

### 1.4 Organization

The organization of this thesis is as follows. Chapters 2 and 3 present the technical approach for addressing the research questions described earlier in this chapter. Specifically, chapter 2 presents the CFD models, flow configurations approximating various physical aspects of flow in NGV-rotor stages with purge flow, and “best practices for implementing NGV-rotor flow computations with purge flows” developed through the course of the thesis. They constitute the research tools for this work and will be referenced throughout the thesis. Next, the framework and methodology for loss estimation and evaluation is presented in chapter 3.

Chapters 4 through 7 present the results of the thesis, with chapter 4 comparing the results between steady and unsteady flow models. The causal links between losses and specific flow processes are delineated in this chapter as well. Chapter 5 focuses on the loss increase from steady to unsteady flow and establishes that it is entirely due to circumferential asymmetry in the purge flow velocity distribution resulting indirectly from the removal of the mixing plane. Chapter 6 describes the use of control volume analysis to estimate the additional loss induced by the circumferentially varying purge flow relative to a circumferentially uniform purge flow for the same purge mass flow rate. Chapter 6 presents the results on the changes in loss generation in response to variation in the purge mass flow rate and swirl. Chapter 7 describes analyses and computations for assessing the concept of endwall contouring as a means of reducing the NGV static pressure field asymmetry (and thus the purge flow velocity asymmetry) for mitigating loss generation from purge-main flow interaction. Lastly, chapter 8 summarizes the thesis and delineates the key findings, followed by suggestions for future work.
Chapter 2

Research Approach

This chapter first introduces the various flow models used to address the research questions - focusing on why they were formulated and their respective strengths and weaknesses. Modifications and improvements made to the models, and the driving reasons behind the changes are also given to establish the implementation process and logic. Lastly, CFD best practice guidelines are delineated, both those specific to certain models and those generally applicable to analysis of purge-main flow interactions.

2.1 Phenomena-Targeted Modeling

The transition from steady to unsteady modeling revealed two primary differences between steady and unsteady flow, and all of the flow feature findings fall into one of these two categories. The first is unsteadiness - the effect of moving rotor blades relative to the stationary nozzle guide vane (NGV). The second is the presence of NGV-induced circumferential variations in the rotor passage that are washed out in the steady case by the mixing plane interface between the stator and rotor passages. Without the mixing plane, circumferential variations from the NGV propagate downstream and influence the purge slot region, which in turn leads to significant circumferential variation in the purge flow injection. This in turn impacts the subsequent viscous shear layer, and its influence continues throughout the rest of the main flow path. These collective effects are responsible for several important flow phenomena described in this thesis, and are the driving force behind changes in loss gener-
ation between steady and unsteady flow situations. Establishing the relative importance of unsteadiness compared to circumferential variation of purge flow is the driving motivation behind the design and formulation of the models presented in this chapter.

The primary focus of this research is on analysis of flow features in a hypothetical turbine stage described in the next section and the response in its behavior in unsteady NGV-rotor flow. Due to the complexity of that flow, a number of simplified computational models were designed and formulated so that the results can be used to isolate specific flow phenomena for assessing their characteristics, and response under different flow conditions. These conditions consist of steady or unsteady flow, and whether or not a mixing plane was imposed between the NGV and purge slot. Presented in the sections after the full sector model are those models that best accomplished this task, and from which results, presented in Chapters 4 through 7, are drawn.

2.1.1 3D Bladed Full Sector

The 3D bladed full sector, depicted in figure 2-1, is the primary flow configuration (i.e. computational domain) used for computations and analyses of unsteady NGV-rotor flow, and is meant to be representative of a hypothetical turbine stage with 32 stators and 64 rotors. It will be referred to as the “full sector” throughout the thesis, so called because it includes a second rotor blade passage to yield a 2:1 ratio of rotor-stator blade number, see figure 2-1 compared to 2-2. Zlatinov’s simulations were performed on a reduced version of the full sector with only a single rotor simulated per passage, figure 2-2, to minimize computing resource requirements. This is possible because the mixing plane approximation in steady simulation delivers a circumferentially uniform flow to the rotor domain. As a result, both rotors see identical incoming flow and produce the same result. Such a simplification cannot be used in unsteady simulation, in which the rotor perceives a circumferentially varying incoming flow, so the full sector is used. The 2:1 ratio was chosen to reduce computational requirements, as it allows a simulation of full sector passage containing one stator blade and two rotor blades, exactly 1/32 of the entire turbine wheel, to represent the performance of the entire stage. Lastly, figure 2-3 depicts the purge flow in-
Table 2.1: Characteristics of a representative turbine stage

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Value</th>
<th>Quantity</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Omega$</td>
<td>2749 [rev/min]</td>
<td>$\sigma_R$</td>
<td>1.37</td>
</tr>
<tr>
<td>$AR_R$</td>
<td>1.11</td>
<td>$\pi$</td>
<td>0.546</td>
</tr>
<tr>
<td>$M_{R,in}$</td>
<td>0.785</td>
<td>$\Lambda$</td>
<td>0.412</td>
</tr>
<tr>
<td>$\Psi$</td>
<td>2.17</td>
<td>$\Phi$</td>
<td>0.65</td>
</tr>
<tr>
<td>$T_{purge}/T_{main}$</td>
<td>0.6</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The boundary conditions are defined as follows. The boundary conditions at the inlet upstream of the stator (NGV) domain consist of a specified spanwise profile in stagnation pressure, flow angle, and stagnation temperature profile. The outlet downstream of the rotor domain is defined by a spanwise profile in static pressure. Purge flow is injected at the hub purge slot, as shown in figure 2-3, with the mass flow rate, flow direction, and stagnation temperature specified. Circumferentially periodic boundary conditions are applied on the sides of the domain to allow the single full sector to be representative of the entire wheel. The NGV-rotor domain interface is a simple fluid-fluid connection, and the nature of the connection varies with the simulation in question - the two domains are connected differently depending on whether steady, unsteady, or frozen rotor analysis is being undertaken. Lastly, in the case in which no purge flow is injected, the purge slot is removed entirely.

Purge Flow Parameters

Four main purge flow parameters were evaluated, though two of the four were shown to be dominant. The first is purge mass flow fraction - the rate of mass ejected from the purge slot made non-dimensional by the main flow mass flow rate. The purge mass flow rate ranges between 0% of the main flow mass flow rate and 2% of the main mass flow rate. The second is purge flow swirl, the amount of circumferential velocity added to the purge flow prior to injection. The value is made non-dimensional by the rotor rim speed. The injected swirl varies from 0% to 150%, where 150% is the approximate value of the main flow
Figure 2-1: Full sector computational mesh - used in all full sector unsteady flow calculations and analyses of NGV-rotor flow
Figure 2-2: Single rotor passage computational mesh - used in Zlatinov’s steady flow analysis

Figure 2-3: Diagram of purge duct in hypothetical turbine stage
circumferential velocity near the hub. The final two parameters are of lesser importance, specifically the purge duct angle ($\phi$) and the purge duct width ($d_w$). These two parameters are depicted in figure 2-4 for clarity.

Figure 2-4: Purge duct schematic highlighting geometric parameters

**Steady State Flow Model**

With the new full sector model and associated mesh presented in figure 2-1, steady state calculations based on a mixing plane approximation were first implemented for assessment against Zlatinov's analysis of purge flow - main flow interaction using the single passage model depicted in figure 2-2. This steady simulation both provides a baseline for later comparison and serves as initial conditions for unsteady NGV-rotor stage simulation. The steady simulation is set up with boundary conditions as described in section 2.1.1, with the interface between stator and rotor set as a mixing plane. This plane mixes out all circumferential non-uniformities present upstream of the interface, so that the flow delivered to the rotor is circumferentially uniform but with total mass, momentum and energy identical to flow upstream of the mixing plane. As a result, the position of the rotor relative to the
NGV is unimportant, as the mixing plane ensures that the rotor is presented with circumferentially uniform flow. In doing this, the mixing plane also creates a significant amount of mixing loss instantaneously.

**Unsteady NGV-Rotor Model**

The boundary conditions for the unsteady simulation are unchanged from the steady simulation to ensure consistent comparison between the two. The unsteady simulation removes the mixing plane that was downstream of the NGV and upstream of the purge slot and rotor, so that the effects of circumferential variations from the NGV are accounted for in the NGV-rotor stage performance quantification. In this case, the position of the rotor relative to the NGV is important, and thus the rotor domain is moved, relative to the NGV, in the numerical simulation.

**Frozen Rotor Simulation**

Frozen rotor simulation is a variant of steady simulation, notably different in that it removes the mixing plane but the rotor’s position, relative to the stator, is fixed. This simulation is useful primarily in that it allows circumferential non-uniformities from the stator to influence the downstream flow, most notably the purge flow, as described earlier in section 2.1, without including unsteady effects. This facilitates analysis to identify the individual role of the two primary effects found in unsteady simulation: unsteadiness and circumferential asymmetry.

**Endwall Contouring**

The endwall contouring is designed to reduce loss driven by purge flow circumferential variation, the basic idea being to reduce the circumferential variation in the purge flow injection and velocity associated with the NGV pressure field. Two different contourings were conceived, one on the rotor side of the purge slot, the other on the stator side of the purge duct. The idea for the rotor endwall contouring was first assessed using a modified version of the bladeless annular case (figure 2-5b) to be described in section 2.1.3, and
compared to the unmodified endwall in figure 2-5a. Note that this is an axisymmetric contouring.

Figure 2-5: Circumferential view of rotor endwall modification to bladeless annular flow configuration on the right to compare with standard (i.e. baseline) configuration on the left.

Similar to the rotor endwall contouring, the stator endwall contouring was designed to reduce the induced circumferential variation in the purge flow - main flow interaction. The stator endwall contouring is depicted in figure 2-6, with the modified regions shown in red, and the circumferential profile of the modified endwall (figure 2-7b) is compared to the unmodified endwall in figure 2-7a. Compared to the rotor endwall contouring, the stator endwall contouring requires a smaller net change to the endwall, as it is not axisymmetric. There is an additional consideration, however, in that the modification is intended to attenuate an effect due to the NGV but must do so without impacting its performance. Calculations for sizing the necessary change, and ensuring it would not impact the stator performance are laid out in section 7.2.2.

2.1.2 Two-Dimensional Axisymmetric Model

The first simplified model is a two-dimensional slice from an axisymmetric annulus representing the sector that eliminates any circumferential variation and contains no blades (figure 2-8). This is effectively a 2D duct with periodic boundary conditions on the sides with a single hub purge slot into which purge flow is introduced. The purge flow can vary in both mass flow rate and swirl, the two parameters Zlatinov found to be of significance in
Figure 2-6: Asymmetric stator endwall contouring - humps follow downstream of the blades.

Figure 2-7: Circumferential view of stator endwall modification to bladeless annular flow configuration on the right to compare with standard (i.e. baseline) configuration on the left.
his steady flow analysis [1], as well as in an unsteady NGV-rotor flow environment (as will be elucidated in Chapter 6). In this simplified flow model, stagnation condition and flow direction are specified at the inflow boundary while static pressure is defined at the outflow boundary. Purge flow is prescribed as a mass flow rate with a specified flow direction at the purge slot. To include unsteady effects, the outlet pressure condition was set to oscillate sinusoidally around the steady outlet pressure value at the blade passing frequency. Figure 2-8 shows the original axisymmetric mesh that was well suited for the steady case (but shown to be inadequate for unsteady flow situation of interest here). In contrast, figure 2-9 depicts the revised mesh, with substantially higher mesh density and reduced passage length (primarily to reduce computational resources needed). The 2D case is designed to analyze the impact of unsteadiness (with no circumferential variation in purge flow) on the formation of a viscous shear layer between the main flow and injected purge flow.

Figure 2-8: Original 2D mesh designed by Zlatinov - proved inadequate for unsteady simulation and was supplanted by the mesh in figure 2-9

2.1.3 3D Bladeless Annular Model

Examination of circumferential flow variation and resultant flow phenomena are critical for understanding unsteady purge flow - main flow interaction, and specifically how they differ
Figure 2-9: Revised 2D mesh - has substantially higher density, but is shortened to reduce computational requirements
from steady state situations. This is first laid out in the findings in section 1.3, and later in section 2.1. The viscous shear layer is the single most important flow feature introduced by purge flow as it is a key source of loss generation; as such the bladeless annular case was designed to assess the effect of circumferential variation in purge flow on the viscous shear layer. As shown in figure 2-10, it consists of a profile somewhat similar to the 2D situation rotated by 11.25°, to match the circumferential extent of the full sector. This is illustrated in figure 2-11, in which the bladeless annular case (green) is overlaid on the full sector (gray). Mass flow and flow direction define the inlet boundary condition, which is taken directly from the rotor/stator interface of the full sector solution, just upstream of the purge slot. The outlet condition matches the average pressure seen prior to the rotor blades, while the unsteady simulation has the back pressure oscillate sinusoidally. In doing this, the bladeless annular model simulates the mixing out of the viscous shear layer without the presence of blades.

Several adaptations were made to the 2D profile of the bladeless annulus relative to the earlier 2D model. Due to the nature of CFX’s solver (which is designed for 3D flow), and modifications in the boundary conditions, the high mesh density necessary for convergence under unsteady simulation in the original 2D mesh could be reduced significantly. This allowed a much larger domain and thus significant additional space for the development of purge flow interaction with main flow. Second, the radius of the hub and shroud had to be modified as the boundary conditions taken from the full sector included finite radial velocity. The first version of the bladeless annular mesh had a hub and shroud of constant radii which led to non-representative flow behavior. Consequently the revised version features a contoured shroud to match the full sector’s shroud and adjusts the hub to maintain the effectively constant cross-sectional area seen in the full sector case.

**Bladeless Annular Model - Endwall Shaping**

As mentioned in section 2.1, circumferential flow variation induced by the NGV results in a circumferentially varying purge flow entering the turbine main flow path. This effect will be shown in Chapters 5-7 to be undesirable, and modifying the rotor endwall offered a potential for reducing it, following the technical arguments laid out in Chapter 7. The 2D
Figure 2-10: Circumferential view of bladeless annular mesh
Figure 2-11: Bladeless annular mesh overlaid with 3D full sector - figure depicts the effective physical location of the bladeless annular model
profile of this contouring compared to the unmodified bladeless annular model 2D profile is depicted in figure 2-5b.

Modifying the stator endwall also presents a possible method of NGV-induced circumferential flow variation reduction, though primarily through a different mechanism, as described in Chapter 7. It also may be the preferable option, as it could be accomplished through a substantially smaller net change in the endwall in the full sector case. Unfortunately, due to modeling constraints, the asymmetric aspect of this concept could not be created in the bladeless annular case, and an axisymmetric version was implemented instead. The 2D profile of this axisymmetric modification is depicted in figure 2-7.

2.2 CFD Simulation Solver Specifications and Guidelines

All calculations were performed using ANSYS CFX version 14.5 using structured meshes. The axisymmetric grid contains 160,000 nodes and results in residuals less than $10^{-4}$ upon convergence. The 3D bladeless annular mesh contains between 1.50 and 1.62 million nodes depending on variant and results in maximum residuals less than $10^{-5}$ upon convergence. The full sector passage mesh contains 1.37 million nodes, with residuals on the order of $3 \times 10^{-3}$ and RMS residuals less than $10^{-4}$ upon convergence. Convergence is defined by a combination of acceptably low residuals and periodic signals from monitor points. The temporal resolution for all simulations was 40 timesteps per period, where a period corresponded to a single blade passing in the axisymmetric model, or a full sector passing in the full sector passage and bladeless annular models. The turbulence level was set to 10%.

2.2.1 Steady State Calculation

The steady simulation CFD solver settings are as follows. The advection scheme is set to high resolution. The turbulence numerics are first order. The timescale factor is 1.0. The turbulence model is the shear stress transport $k-\omega$ model based on turbulence kinetic energy and kinematic eddy viscosity equations.
2.2.2 Unsteady Simulation

The timestep for unsteady flow simulation was chosen to be $1/40$ of a blade passing period, and unsteady simulations were started using fully converged steady simulations as initial conditions. Unsteady simulations were run for a minimum single wheel passing - 32 sector passings, before being assessed for stable periodic behavior that reflects equilibrium unsteady solution. Longer time steps, both $1/20$ and $1/15$ of a blade passing period, appeared suitable for use in the 3D full sector, but were demonstrably insufficient for the 2D axisymmetric and 3D bladeless annular models. For consistency, all unsteady simulations were implemented with a timestep equal to $1/40$ of a blade passing. The transient scheme is a second order backward Euler scheme. All other settings were unchanged from the steady simulation.

2.3 Summary

This chapter presented the various levels of physical modeling that were used to address the research questions posed in Chapter 1. The required attributes of CFD needed to produce an adequate computed flow field for physical interpretation are delineated. For reference, the following table contains all of the computational models and flow configurations used in CFD in this thesis.
Table 2.2: Table of computational models and flow configurations used in CFD, uniform denotes circumferentially uniform while varying denotes circumferentially varying

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Simulation Type</th>
<th>Circumferentially uniform or varying</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full Sector</td>
<td>Steady</td>
<td>Uniform</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Unsteady</td>
<td>Varying</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Frozen Rotor (steady)</td>
<td>Varying</td>
<td>A steady model, removes mixing plane from rotor/stator domain boundary, allowing for circumferential variation</td>
</tr>
<tr>
<td>3D Bladeless Annular (B.A.)</td>
<td>Steady</td>
<td>Uniform</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Unsteady</td>
<td>Uniform</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Varying</td>
<td>Varying</td>
<td></td>
</tr>
<tr>
<td>3D B.A. with Rotor Endwall Modification</td>
<td>Unsteady</td>
<td>Uniform</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Varying</td>
<td>Varying</td>
<td></td>
</tr>
<tr>
<td>3D B.A. with Stator Endwall Modification</td>
<td>Unsteady</td>
<td>Uniform</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Varying</td>
<td>Varying</td>
<td></td>
</tr>
<tr>
<td>2D Annulus</td>
<td>Steady</td>
<td>Uniform</td>
<td>As a 2D model, it cannot possess circumferential variation</td>
</tr>
<tr>
<td></td>
<td>Unsteady</td>
<td>Uniform</td>
<td></td>
</tr>
</tbody>
</table>
Chapter 3

Entropy Based Loss Accounting

3.1 Introduction

The overarching goal of this thesis is to quantify loss generation associated with purge flow and main flow interaction in axial turbines in an unsteady NGV-rotor stage flow environment. Following this, suggestions for mitigating the opportunity for loss generation are pursued. Chapter 2 presented flow models at various levels of approximation to facilitate the identification of loss generation linked to specific flow effects.

This chapter describes the framework for quantifying loss generation and for establishing the link between loss production and the responsible flow process. Specifically, the metric used for loss quantification is irreversible entropy generation. Generation of entropy in irreversible processes provides a measure of lost potential to do work, as has been established in literature [1] and can directly be cast in terms of turbine efficiency debit. Denton’s 1993 IGTI paper was instrumental in understanding and developing loss estimation and quantification tools used in this thesis, in particular the adaptation to loss estimation and quantification in unsteady flow simulation [13].

3.2 Entropy Based Loss Accounting

Entropy change in fluid flow is described by equation 3.1.
\[ \iint_V \rho \frac{DS}{Dt} \, dV + \int_A \frac{q_{in}}{T} \, dA = \iint_V S_{gen}'' \, dV \quad (3.1) \]

It is the right hand side term that is of greatest interest here, and significant use will be made of entropy generation on a volumetric basis, specifically the volumetric entropy generation rate, denoted as \( S_{gen}'' \). A volumetric rate quantity is particularly appealing because it can be integrated over defined volumes to establish the entropy generation, and hence the loss, in individually defined volumes. Furthermore, when appropriately scaled by temperature, it becomes a measure of power dissipation due to determined losses, a useful metric that can also be used to calculate overall efficiency.

Even volumetric entropy generation rate can be further refined, however, as entropy generation can be due to either viscous (\( S''_{visc} \), equation 3.3) or thermal (\( S''_{therm} \), equation 3.4) effects, as stated in equation 3.2.

\[ S_{gen}'' = S''_{visc} + S''_{therm} \quad (3.2) \]

\[ S''_{visc} = \frac{1}{T} \mathcal{T}_{ij} \left( \frac{\partial v_i}{\partial x_j} \right) = \left( \frac{\mu + \epsilon}{T} \right) \left( 2\dot{e}^2 - \frac{2}{3} (\nabla \cdot \mathbf{u}) \right) \quad (3.3) \]

\[ S''_{therm} = \frac{k_{eff}}{T^2} \left( \frac{\partial T}{\partial x_i} \right)^2 \quad (3.4) \]

Categorizing \( S_{gen}'' \) into \( S_{visc}'' \) and \( S_{therm}'' \) is useful because losses generated by \( S_{therm}'' \) are not useful for the stated goal of assessing the flow to improve turbine performance. The details of the argument not to include \( S_{therm}'' \) are described in Chapter 3 of Zlatinov's thesis [1], but the essence of it is that \( S_{therm}'' \) comes about from failing to extract work from the temperature difference between main and purge flows. Penalizing the turbine for not taking advantage of this temperature difference is to penalize a turbine stage for not fulfilling the role of a heat engine. Additionally, \( S_{therm}'' \) is controlled by two parameters - purge mass flow and temperature. Minimizing purge mass flow is already known to be desirable - but it must be sufficiently high so as to prevent ingestion, and any design improvement can do little to change the thermal loss from a set rate of purge mass flow. The second variable,
temperature, is a function of where the purge air originates from within the compressor, and thus not a variable parameter for the purpose of purge slot design.

$S'''_{\text{visc}}$ has two primary uses in the research described in this thesis. First, contours depicting it can be used to track or determine entropy generation and thus loss from any viscous source in the domain. Second, improved implementation of this quantity since Zlatinov’s work has led to it being generally accurate quantitatively in all regions, so it can now be integrated over defined volumes to output viscous loss production. Specifically, Zlatinov noted a discrepancy in regions with large gradients, but this has been almost entirely resolved. In practice, $(S'''_{\text{visc}}) \times T$ is scaled by temperature to give an expression for volumetric power dissipation due to viscous sources, a useful metric. In this thesis, $(S'''_{\text{visc}} \times T)$ is cast as a non-dimensional coefficient: $(S'''_{\text{visc}} \times T)/(\Delta h_{\text{volumetric}})$. The denominator, $(\Delta h_{\text{volumetric}})$, is the average volumetric change in stagnation enthalpy over the rotor domain. The coefficient measures the local volumetric power loss due to viscous sources compared with the average volumetric stagnation enthalpy extraction rate by the turbine, essentially a metric that describes the relative significance of loss generation in the local flow. A value of 5, for example, indicates that on a volumetric basis the local flow element is responsible for 5 times more viscous loss than the average volumetric change in stagnation enthalpy across the domain.

### 3.3 Viscous Lost Work

The second primary loss metric used is viscous lost work ($W_{\text{visc-loss}}$), which is primarily used for calculating loss profiles and overall efficiency for comparison between cases. Viscous lost work has two advantages over simply integrating $S'''_{\text{visc}}$. First, the numeric is exactly accurate in all regions, unlike $S'''_{\text{visc}}$, which while greatly improved from Zlatinov’s work, does have small error in regions with high gradients. Thus in instances in which the unique attributes of $S'''_{\text{visc}}$ are not necessary, primarily evaluation of localized loss generation, viscous lost work is the preferred method for calculating loss. Second, viscous lost work can capture the loss impact of the mixing plane, a discontinuity in the flow that integrating a volumetric variable will not account for. These two factors make viscous lost
work a more suitable measure for tracking overall loss behavior across the stage, whereas $\dot{S}_{visc}^{''''}$ is more useful for examining or isolating specific loss sources.

### 3.3.1 Viscous Lost Work in Non-Uniform Streams

The figure in this subsection (figure 3-1) is taken from [1], along with an abridged description of the underlying analysis. Readers looking for a more complete derivation and explanation should examine section 3.1 of said work. The goal of the viscous lost work method is to quantify the amount of work lost to viscous sources in the expansion of two non-uniform streams.

![T-s diagram of multi-stream expansion illustrating the concept of work-averaged pressure.](image)

**Figure 3-1:** T-s diagram of multi-stream expansion illustrating the concept of work-averaged pressure.

Before going further the two non-uniform streams must be replaced with equivalent
uniform streams. The process by which this is done is a work-averaging method formulated by Cumpsty and Horlock in [14], chosen such that the averaged streams produce the same work output as those they are replacing. This process is depicted in figure 3-1. The higher pressure streams can be isentropically expanded to a stagnation pressure \( P_{t2}^{\text{wa}} \) to provide the exact amount of work necessary to compress the second stream to the same stagnation pressure. This is an example of the "work-averaging" described in [14]. The appropriate equation is below, equation 3.5.

\[
P_i^{\text{wa}} = \left( \frac{\int T_i \, dm}{\int \frac{T_i}{T_i^{\text{wa}}} \, dm} \right)^{\frac{\gamma - 1}{\gamma}} \tag{3.5}
\]

The temperature is mass averaged because it fixes the state of the averaged flow while maintaining conservation of energy. Doing so is an irreversible thermal mixing of the two streams at constant pressure. The entropy created in this process is denoted as \( \Delta s_{\text{therm}} \), and this removes the work that otherwise could be extracted via a heat engine. The resulting ideal expansion work of the two streams is presented in equation 3.6.

\[
w_{\text{expand}} = c_p T_{t1}^{\text{ma}} \left( 1 - \left( \frac{P_{t2}^{\text{wa}}}{P_{t1}^{\text{wa}}} \right)^{\frac{\gamma - 1}{\gamma}} \right) \tag{3.6}
\]

However, in any physical process there will be viscous generation of entropy, denoted as \( \Delta s_{\text{visc}} \). This will correspond with reduced potential for work, expressed in equations 3.7 and 3.8.

\[
w_{\text{visc-loss}} = \tilde{T}_{t2} \Delta s_{\text{visc}} = \tilde{T}_{t2} (\Delta s - \Delta s_{\text{therm}}) = w_{\text{expand}} - w_{\text{actual}} \tag{3.7}
\]

\[
w_{\text{visc-loss}} = c_p \left( T_{t2}^{\text{ma}} - T_{t1}^{\text{ma}} \left( \frac{P_{t2}^{\text{wa}}}{P_{t1}^{\text{wa}}} \right)^{\frac{\gamma - 1}{\gamma}} \right) \tag{3.8}
\]

The above provides a consistent method that accounts for viscous losses in turbomachinery while factoring out losses from thermal mixing. The application of the method to CFD in the current thesis is presented in the next subsection.
3.3.2 Current Implementation of Viscous Lost Work Formulation

$W_{\text{visc-loss}}$ is formed by using averages of two quantities over an axial plane: work averaged stagnation pressure ($P_{\text{t,avg}}$, equation 3.5) and mass averaged stagnation temperature at each plane at each timestep. Using equation 3.8 forms a single metric that establishes the work lost to viscous effects from the inlet through that plane on a time average basis. This equation can be applied to a set of axial planes spanning the domain to create profiles of loss due to viscous effects. When applied to the outlet of the domain, it yields the total lost work, which can be combined with the ideal (isentropic) work extracted ($W_{\text{ideal}}$, equation 3.9), to establish overall stage efficiency ($\eta$, equation 3.11). Note that when using the ideal work equation that the "in" quantities must account for both main flow from the inlet and purge flow, weighted by respective mass flow rate.

$$W_{\text{ideal}} = c_p m T_{t-in} \left(1 - \left(\frac{P_{t-out}}{P_{t-in}}\right)^{\frac{\gamma-1}{\gamma}}\right)$$  \hspace{1cm} (3.9)

efficiency debit = $\frac{W_{\text{visc-loss}}}{W_{\text{ideal}}}$  \hspace{1cm} (3.10)

$$\eta = 1 - \frac{W_{\text{visc-loss}}}{W_{\text{ideal}}}$$  \hspace{1cm} (3.11)

3.4 Time Averaging of Unsteady Flow

Time averaging is needed in unsteady analysis, as comparing two unsteady flows on a timestep by timestep basis is often untenable. Additionally, time averaging is necessary for assessments against steady analysis. However, incorrect time averaging can give a false result. For example, when accounting for fluxes, the direct method is to simply mass average the time flow average quantity. However, doing so does not account for variation in mass flow in time and is thus incorrect. Specifically, it does not account for the fact that if the mass flow is varying significantly in time, the averaging should weight the value of the quantity of interest at times with higher mass flow to reflect that.
3.4.1 Evaluating Time-Average Loss

Three methods for quantifying loss generation have been established in this chapter. The following section describes how each is time averaged in unsteady flow.

**Volumetric Entropy Generation Rate**

Time averaging of $S'''$ is comparatively direct. It is on a volumetric basis, not a mass basis, and the volumes are constant in time. Specifically the CFD element volume sizes and shapes are unchanging. So creating a time averaged measurement just requires taking the arithmetic average of $S'''$ at all timesteps.

**Viscous Lost Work**

Time averaging of viscous lost work requires that the values of several terms be obtained at all timesteps, and then combined to form the time average of the overall viscous lost work term. Specifically, the numerator and the denominator of the work averaged stagnation pressure equation (equation 3.5, reproduced here as equations 3.12 and 3.13) must be measured at all timesteps, as must the mass averaged stagnation temperature. Combining these terms to form the viscous lost work (equation 3.8) must be done on a timestep by timestep basis, and only then averaged arithmetically to obtain the accurate time average of the viscous lost work term.

\[
\int T_i dm \tag{3.12}
\]

\[
\int \frac{T_i}{P_i^{\gamma-1}} dm \tag{3.13}
\]

**Control Volume Analysis of Two Stream Mixing**

To appropriately time average the two stream mixing in control volume analysis, the CV analysis is simply performed at every timestep, and weighted by the fraction of overall mass flow represented in that specific timestep.

49
3.5 Attributing Loss to Flow Phenomena

While computational fluid dynamics methods make determining overall quantities such as stage loss or efficiency relatively straightforward, they do little inherently to facilitate attribution of entropy generation to specific flow phenomena. This section provides a brief outline of the procedures used for identifying loss phenomena and attributing proportional responsibility.

3.5.1 Geometric Bounding Method

In addition to determining total lost work, it is useful to know the proportional contribution of each major loss inducing flow feature to overall lost work. To this end, several methods for delineating loss due to different sources within the unsteady full sector have been formulated. The first method combines geometric bounds with a tracking variable, the purge flow passive scalar. The purge flow passive scalar is a variable used for tracking the mass fraction which is purge flow in any given flow element. It operates by starting as a value of 1 at the purge inlet, and diffusing into and through the main flow via viscosity and the transport equation, equation 3.14, where $\sigma_i$ is the proportion of the element that is purge flow.

$$\frac{\partial \sigma_i}{\partial t} + \frac{\partial (\sigma_i U_i)}{\partial x_j} = -\frac{\partial}{\partial x_j}(\sigma_i (U_{ij} - U_j) - \sigma''_i U''_j)$$

(3.14)

This variable allows easy tracking of where the bulk of the purge flow travels after exiting the purge slot. The downside of this method is an inherent level of subjectivity based upon where the boundary is placed between the flow phenomena being delineated. When this data is presented in Chapter 5, it will include a quantification of sensitivity to moving this boundary.

3.5.2 Two Stream Mixing Method

The second method described in this section is a control volume method that takes CFD data as an input to estimate the entropy produced in the mixing of two non-uniform streams;
this would provide an estimate of loss that will be generated in the viscous shear layer.

Specifically, the method takes information about the flow from the purge slot, near-hub, main flow just upstream of the purge slot, and in the mixed-out flow well downstream; these are depicted in figure 3-2. The data is obtained by averaging radially or axially on planes over the regions of interest to form circumferential profiles of the relevant quantities - namely all three velocity components, density, and mass flow. The equation used is equation 3.15, and is based on the work of Young and Wilcock [15]. The method breaks the main and purge flows into an equal number of streamtubes (with circumferential boundaries) and pairs them up by location. Then each pair of streamtubes (one of main flow and one of purge) is mixed as described in equation 3.15.

\[
\frac{\dot{W}_{\text{loss}}}{m} \approx \frac{m_{st}}{m_{\text{total}}} \left( (v_{x,\text{main}} - v_{x,\text{purge}})^2 + (v_{r,\text{main}} - v_{r,\text{purge}})^2 + (v_{\theta,\text{main}} - v_{\theta,\text{purge}})^2 \right) \quad (3.15)
\]

where \(m_{st}\) is the mass flow rate within the purge flow streamtube under examination, \(m_{\text{total}}\) is the total purge mass flow rate, \(v_{\text{purge}}\) is the purge flow velocity, and \(v_{\text{main}}\) is the main flow velocity, with \(x, r, \) and \(\theta\) subscripts indicating components in cylindrical coordinates.

The positioning, shape, and size of the averaging regions in the bladeless annular case are depicted in figure 3-3, and the equivalent figure for the full sector is figure 3-4.

A limitation is that this method is useful only for comparison - the loss numbers it produces are subject to simplifying assumptions so they are less accurate than results obtained from equations 3.3 and 3.8 seen earlier in this chapter. Rather, it estimates the amount of loss from mixing to a designated end state. By comparing two such numbers, the relative change in loss from viscous shear layer mixing can be estimated.

### 3.6 Summary

This chapter summarized the equations and methods used for loss estimation and quantification. Each of these methods has strengths and weaknesses. \(S''''_{\text{visc}}\) is useful for locating loss regions, but less accurate for creating overall loss profiles due to assumed discontinuities.
or presence of high flow gradients in the discrete volumes. Viscous lost work is suitable for general loss computation, but less helpful in attributing loss to flow phenomena. Lastly, a control volume method was developed for two stream mixing aimed specifically at estimating the loss generated in the mixing out of main and purge flows.
Figure 3-3: This figure depicts the locations from which main flow (red), purge flow (blue), and mixed-out flow (magenta) data was taken in the bladeless annular case to estimate the losses from mixing out of the shear layer associated with purge flow introduction.

Figure 3-4: This figure depicts the locations from which main flow (red) and purge flow (blue) data was taken from the full sector case to estimate the losses from the mixing out of the shear layer associated with purge flow introduction.
Chapter 4

Unsteady Flow and its Time Averaged Impact

The focus of this chapter is evaluating lost work and entropy generated through the stage, hence the overall performance of the turbine stage. The computed results based on the approach presented in chapter 2 and 3 are analyzed and interrogated to assess the impact on turbine performance for purge flow with varying mass flow rate, swirl, injection angle, and purge slot width.

The organization of this chapter is as follows. First, the parametric trend in loss generation from unsteady flow is assessed against that from steady state flow (based on mixing plane approximation) for representative variation in purge flow parameters. This is followed by establishing the link between loss generation and specific flow regions and features. This link will be later used to determine the mechanism responsible for the difference in loss generation between unsteady flow and assumed steady state flow situation.

There are two primary regions in which purge flow related loss is generated. The first is dubbed the “viscous shear layer” (VSL), and denotes all losses generated through the mixing of the purge flow with the main flow. The second is called the “secondary flow region” (SFR), and represents additional losses generated by the introduction of purge flow not due to the mixing of purge flow and main flow. These are primarily effects that are tied to the presence of blades in the flow path, such as the passage vortex. Tip clearance loss is excluded from this category. These regions are depicted in figure 4-5.
Lastly, a remark on presentation of unsteady data: unsteady loss data is always presented on a time-average basis, as described in chapter 3, for assessment against the steady result. Velocity or pressure profiles, in contrast, are shown using a single or multiple distinct time instants, as the intent with these figures is typically to show variation with time.

4.1 Net Loss Trends

The net loss produced over the entire NGV-rotor flow domain for purge mass flow between 0% and 2% of the main flow mass flow rate, and swirl between 0% and 150%, is presented in table 4.1. The loss data presented is normalized by the steady loss for the baseline case with no purge flow injected and the purge slot removed. The purge mass column denotes the amount of purge mass flow as a ratio of the entering main flow. The purge swirl is the circumferential velocity of the injected purge flow made non-dimensional by the rotating hub speed, expressed as a percentage.

The basic trends shown in this table are similar to those documented by Zlatinov, namely that loss increases with increasing purge mass flow rate, and that it decreases as the flow is swirled. These trends hold for both steady and unsteady flows. Comparing unsteady and steady flows, the unsteady flow produces consistently higher loss. Lastly, note that the difference in loss between the steady and unsteady flows jumps between the 0% and 0.5% purge mass flow rate cases, from 4% to over 8%. This is significant as it suggests that there is a specific feature of unsteady flow that causes purge flow to increase loss generation.

4.1.1 Viscous Lost Work Profiles

Three plots are presented here to give a basic depiction of how losses are generated through the stage for purge mass flow between 0% and 2% of the main flow mass flow rate, and swirl between 0% and 150%. The loss metric is cumulative viscous lost work generated over the stage, as described in section 3.3, using equations 3.5 through 3.8. All unsteady data is time-averaged and presented side-by-side with its steady counterpart for comparison. The first is figure 4-1, which presents axial variation in loss profiles under varying purge flow
<table>
<thead>
<tr>
<th>Purge mass flow</th>
<th>Purge Swirl</th>
<th>Steady Loss</th>
<th>Unsteady Loss</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0</td>
<td>1.0</td>
<td>1.040</td>
</tr>
<tr>
<td>0.5</td>
<td>0</td>
<td>1.042</td>
<td>1.127</td>
</tr>
<tr>
<td>1.0</td>
<td>0</td>
<td>1.085</td>
<td>1.160</td>
</tr>
<tr>
<td>2.0</td>
<td>0</td>
<td>1.166</td>
<td>1.225</td>
</tr>
<tr>
<td>2.0</td>
<td>50</td>
<td>1.089</td>
<td>1.168</td>
</tr>
<tr>
<td>2.0</td>
<td>100</td>
<td>1.051</td>
<td>1.132</td>
</tr>
<tr>
<td>2.0</td>
<td>150</td>
<td>1.025</td>
<td>1.108</td>
</tr>
</tbody>
</table>

rate. In the figure, notice that the total loss generation (indicated by the right-most points) increases approximately linearly with mass flow rate, and that the unsteady flow loss is always higher by the end of the domain.

As noted earlier, there is a significant difference between the unsteady 0% purge mass flow case and any of the others, as shown by the divergence between the unsteady loss profiles just upstream of the rotor leading edge (at x=0). The divergence begins at the purge slot. The discontinuity in the steady profiles comes about from the presence of the mixing plane (which produces a flow in which circumferential non-uniformity is mixed out). The equivalent mixing loss is generated in the unsteady cases over the rest of the rotor domain.

The second figure (figure 4-2) presents loss profiles with varying swirl but with a constant 2% purge mass flow rate. As in figure 4-1 for purge mass flow rate variation, notice that the unsteady flow equivalent of any steady flow situation generates more loss by the end of the computational domain. However, the relation between loss and swirl is clearly non-linear, and will be explored further in section 6.2.

### 4.1.2 Removal of Purge Slot

In section 2.1.1 it was noted that in the case that no purge flow is injected, the purge slot is entirely removed from consideration. This is done to prevent the introduction of new confounding effects, which have the potential to generate loss that should not be attributed to the 0% mass flow case. Figure 4-3 depicts non-dimensional $T^\text{visc}$ contours on the full
Figure 4-1: Full sector model: cumulative viscous lost work profile from inlet to exit boundary for varying purge mass flow rate. The vertical black lines indicate the purge slot location.
Figure 4-2: Full sector model: cumulative viscous lost work profile from inlet to exit boundary for varying purge flow swirl. The vertical black lines indicate the purge slot location.

sector purge flow with and without the slot present. The figure shows the production of a viscous shear layer from flow exiting the slot.

This phenomena increases the loss produced in unsteady flow, as shown in figure 4-4, which compares 0% purge mass cases with and without the purge slot, under steady and unsteady simulation. The steady flow loss profile is effectively unchanged by the slot, with the minor difference too small to be measurable. The unsteady simulation, however, produces markedly less loss with the slot removed.

4.2 Loss Breakdown by source

The focus of this section is on attributing the loss identified in the previous section to specific flow features or regions. This is important because it serves as the first step in
Figure 4-3: Full sector model - effect of purge slot with zero mass flow rate on loss generation. Top figure depicts loss contour with the purge slot present, bottom figure shows the contour with the purge slot removed.
Figure 4-4: Full sector model - cumulative viscous lost work profile from inlet to exit boundary for 0% purge mass flow rate. The vertical black lines indicate the purge slot location.
determining the mechanisms behind the additional loss caused by purge flow in the NGV-rotor unsteady flow environment, and the relative significance of each mechanism. We begin by first presenting the basic loss delineation method used. The relevant loss sources will then be described and explained in turn in the sections that follow.

The delineation is accomplished via the method of geometric bounding, as described in section 3.5.1. This method uses a combination of geometric bounds and purge flow tracking via a passive scalar to determine regions in which purge flow is likely to have impact, then divide the greater purge flow volume into regions by the dominant feature present. The resulting division is shown in figure 4-5, in which the orange isovolume marks the region associated with the viscous shear layer, and the blue isovolume the passage secondary flow region.

![Figure 4-5: Definition of regions each with specific loss generating flow features: viscous shear layer and passage secondary flow](image)

The process of attributing loss to these flow regions and thus their underlying flow features relies upon the entropy generation rate due to viscous sources, scaled by temperature, $T \dot{S}_{\text{visc}}$. First, $T \dot{S}_{\text{visc}}$ is integrated over the relevant region in the flow of interest. That exact same geometric volume is imported into the baseline flow case (no purge flow), and
the same integration is performed. The result is the difference in $T\delta''_{visc}$ for which the interaction of purge flow with main flow is responsible. The difference in the two numbers expresses an approximate power dissipation due to viscous effects that can be attributed to purge flow injection. By comparing these individual dissipations to the overall change in dissipation in the sector, the relative contributions of the loss regions can be determined. The result of this process are presented in table 4.2, with MF specifying purge mass flow rate as a percentage of main mass flow rate, S or US denoting steady or unsteady flow, VSL denoting the viscous shear layer, and SFR representing the secondary flow region.

For steady flow, the computed results are in good accord with those of Zlatinov described in reference [1] which used a somewhat different method for categorizing the different sources of loss generation. Present method yielded an approximate breakdown of 55%, 25%, and 20% proportional loss contribution (or total loss) from the viscous shear layer, secondary flow region, and other, respectively, all for a 1.5% purge flow case without swirl. In reference [1], other loss sources mostly consist of tip clearance loss. There are noticeable differences in the contribution associated with the viscous shear layer, but within the combined error bounds of the methods used. The following subsections will analyze the specific regions, their trends, and any underlying loss sources.

### 4.2.1 Viscous Shear Layer

The viscous shear layer is the flow feature produced by the interaction of the purge flow leaving the purge duct and the main flow, and subsequent mixing through the stage. It is the largest single loss source introduced by purge flow, and its importance increases in

---

**Table 4.2: Full Sector Loss Source Attribution**

<table>
<thead>
<tr>
<th>MF</th>
<th>Swirl</th>
<th>S/US</th>
<th>V.S.L. (%)</th>
<th>Error</th>
<th>S.F.R. (%)</th>
<th>Error</th>
<th>Remainder (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0%mf</td>
<td>0%</td>
<td>S</td>
<td>46</td>
<td>3</td>
<td>29</td>
<td>2</td>
<td>25</td>
</tr>
<tr>
<td></td>
<td></td>
<td>US</td>
<td>52</td>
<td>3</td>
<td>32</td>
<td>2.5</td>
<td>16</td>
</tr>
<tr>
<td>2.0%mf</td>
<td>0%</td>
<td>S</td>
<td>47</td>
<td>3</td>
<td>30</td>
<td>2</td>
<td>23</td>
</tr>
<tr>
<td></td>
<td></td>
<td>US</td>
<td>55</td>
<td>3</td>
<td>33</td>
<td>2</td>
<td>12</td>
</tr>
<tr>
<td>2.0%mf</td>
<td>100%</td>
<td>S</td>
<td>46</td>
<td>3</td>
<td>33</td>
<td>3</td>
<td>16</td>
</tr>
<tr>
<td></td>
<td></td>
<td>US</td>
<td>41</td>
<td>3</td>
<td>44</td>
<td>3</td>
<td>15</td>
</tr>
</tbody>
</table>
unsteady flow, as table 4.2 suggests that it is also the dominant source of the additional loss produced in unsteady flow.

**Purge Flow Blocking**

A critical feature regarding the interaction between purge flow and main flow that constitutes the viscous shear layer is that in unsteady flow it is no longer circumferentially uniform. The removal of the mixing plane leaves the flow exiting the stator with significant circumferential non-uniformities. Most significantly, the stator pressure field extends downstream and interacts with the purge duct and emerging purge flow, leading to circumferential variations in the purge flow distribution. In the area of maximum pressure influence from the NGV, this effect is sufficient to block purge flow ejection entirely. This phenomena is depicted for the 2% purge mass flow case in figure 4-6, which shows a contour of \( \frac{P_{\text{main}}}{P_{\text{stagnation-purge}}} \) at 1% span. The figure only distinguishes regions above a value of 1, that is, regions in which the static pressure of the main flow exceeds the stagnation pressure of the purge flow. Any such region over the purge duct will lead to the purge flow stagnating against the main flow and failing to enter the main flow path. A comparison between steady and unsteady purge flow velocity profiles is shown in figure 4-7, which shows the impact of \( P_{\text{main}} > P_{\text{stagnation-purge}} \) has on the flow at a representative time instant. A dotted line is extended from the average radial velocity of the steady purge flow, to elucidate the circumferentially varying radial velocity for the unsteady flow situation. The region corresponding to \( P_{\text{main}} > P_{\text{stagnation-purge}} \) is indicated by the yellow oval - notice that the radial velocity is near zero and that the circumferential velocity increases to a value closer to the main flow. The figure shows that significant circumferential non-uniformities in velocity are introduced. Continuity requirements necessitate the existence of circumferential locations with higher purge flow ejection velocity; this is shown by the region around circumferential position 4, where the radial velocity is above the steady average. Thus there are regions of high purge flow and low to zero purge flow as the purge flow must satisfy flow continuity (i.e. same mass flow at purge exit as at purge slot inlet). The change in the circumferential distribution of purge flow ejection with time is illustrated for 4 equally spaced timesteps over a blade passing period in figure 4-8. The contours are for the pas-
sive scalar distribution at 1% span - the percentage of flow in a given cell that is purge flow at said position. Notice the clear circumferential non-uniformity, with the region of least purge flow in approximate alignment with the stator location. The circumferential location actually moves slightly (due to influence from the rotor pressure field), between the position that is axially downstream from the stator trailing edge and the position that is streamwise downstream of the stator trailing edge.

Figure 4-6: Near hub pressure plot showing regions of purge flow being blocked for 2% purge mass flow

Furthermore, the region corresponding to $P_{\text{main}} > P_{\text{stagnation-purge}}$ grows when less mass flow is injected by the purge slot. In this situation the injected flow has substantially lower stagnation pressure, and thus a larger proportion of the main flow directly above the slot has static pressure in excess of the reduced purge flow stagnation pressure. This is presented in figure 4-9, which shows the equivalent plot of the results in figure 4-6, only with 0.5% injected purge flow instead of 2.0%. Consequently, the size of the blocked region has increased. Finally, while hot gas ingestion largely falls outside the scope of this thesis
research, the regions where $P_{\text{main}} > P_{\text{stagnation-purge}}$ are the most likely to be susceptible to hot gas ingestion.

![Figure 4-7: Purge flow velocity profile at purge slot exit plane. The x-axis displays the circumferential position in degrees, with the domain equivalent to the full sector passage - 2 rotors and 1 NGV.](image)

### 4.2.2 Bladeless Annular Model

The bladeless annular model, designed to model the viscous shear layer of the full sector in isolation, is used for assessing its trends and behavior in the absence of any flow effects induced by the rotor blades. The inlet boundary condition replicates flow conditions at a plane just after the interface between the stator and rotor, and thus captures the effects of the mixing plane (steady flow), or absence thereof (unsteady flow). As a result, the effect of any circumferential variation in unsteady purge flow is reflected in this model. The outlet condition is a moving circumferential profile of flow variables that reflects the unsteady effects due to the moving rotors. Together, these features should result in a reasonably good approximation of how the viscous shear layer would develop as it proceeds downstream.

This data is presented in the same format as the full sector data, in table 4.3 and figure 4-10. The basic trends observed are consistent with those observed in the full sector - additional purge flow mass increases loss generated while swirling the purge flow reduces it. The scale of the differences between the steady and unsteady flow, along with the differences between cases with different purge flow parameters is substantially higher. The
Figure 4-8: Full sector asymmetry visualization via purge flow tracking - 2% purge mass flow
Figure 4-9: Near hub pressure plot showing regions of purge flow being blocked for 0.5% purge mass flow
difference is higher due to lower loss generated through the stage as a whole because without blades there is no profile loss. It is to be noted that the inlet condition for the bladeless annular model is taken after the mixing plane. Thus the steady bladeless annular model flow has incurred a mixed-out loss whereas the unsteady bladeless annular model flow has not; one should account for this in interpreting the difference between the axial variation in loss computed from the steady and unsteady bladeless annular model flow shown in figure 4-10. In the zero purge flow situation, the two curves (solid black line and discrete point curve) should approach one another as the circumferential non-uniformity in the unsteady flow is mixed out if the mixed-out loss incurred on the steady flow is accounted for.

Lastly, unsteady swirled flow appears to have increased secondary flow loss, but this is not actually the case, but rather a consequence of the loss attribution method. Drawing upon table 4.2 compared to 4.3 and figure 4-10 the share of loss generated in the viscous shear layer increases from steady to unsteady flow, unless the injected purge flow is swirled. However, the loss profile for the bladeless annular model depicts the largest increase, both proportionally and in real terms, for each specific case. This contradiction can be resolved by examining the character of the difference. Note that in figure 4-10 around 1.25 duct heights downstream, this difference in the 2% purge flow case reaches a near asymptotic state, changing only marginally over the rest of the domain (difference grows by less than 10%). In comparison, in the swirled case the difference increases noticeably between 1.25 duct lengths downstream of the inlet and the outlet, increasing by more than 30%. Comparing this result to table 4.2 for the full sector, it can be inferred that the swirled case has the largest proportional loss difference between the steady and unsteady secondary flow region - suggesting that there is a greater proportion of loss resulting from mixing between the main flow and purge flow in the full sector.

### Table 4.3: Viscous Lost Work in Bladeless Annular Model

<table>
<thead>
<tr>
<th>Purge mass flow</th>
<th>Purge Swirl</th>
<th>Steady Loss</th>
<th>Unsteady Loss</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0</td>
<td>1.0</td>
<td>1.13</td>
</tr>
<tr>
<td>2.0</td>
<td>0</td>
<td>2.10</td>
<td>2.29</td>
</tr>
<tr>
<td>2.0</td>
<td>100</td>
<td>1.23</td>
<td>1.44</td>
</tr>
</tbody>
</table>
Figure 4-10: Bladeless annular model - cumulative viscous lost work for varying parameters. The vertical black lines indicate the purge slot location.
4.2.3 Full Sector - Secondary Flow Region

Compared to the viscous shear layer, the secondary flow region is responsible for a smaller proportion of loss except in the situation in which purge flow is swirled. However, the bladeless annular case suggested that the increase in loss generation in the secondary flow region with swirl is due to the viscous shear layer taking a longer distance to mix out in unsteady flow, and thus being accounted for in the secondary flow region. Thus it is not due to additional purge flow loss induced by the blades. With this correction, the change in loss in the secondary flow region between steady and unsteady flow becomes small in all cases. There is a single new effect described in the next subsection that is responsible for what difference there is.

Hub-Corner Separation

There is one significant new loss-generating feature in secondary flow region in unsteady flow, a small but measurable region of loss at roughly 50% chord at the rotor-hub corner on the suction side. This appears to be a region of hub-corner separation, and the relevant region is depicted in steady flow in figure 4-11 and unsteady flow in figure 4-12. In the contour plot of time-average $T'S''_\text{visc}$ for unsteady flow, the separation region is indicated by the black circle. The figures show both streamlines of the relevant flow, and a non-dimensional $T'S''_\text{visc}$ contour to show loss generation rate. In the steady flow the streamlines are clearly attached as they traverse up the rotor, and relatively little loss is generated. In the unsteady flow, the streamlines appear to separate from the rotor, accompanied by a significantly higher loss generation rate in the region.

In unsteady flow with 0% swirl purge flow, this separation is responsible for 1-1.5% of the loss introduced by unsteady flow. Under swirled purge flow this increases to 2-4% of loss introduced by unsteady flow. While these numbers are relatively small compared to the loss associated with unsteady flow viscous shear layer, the finite extent of the separation region makes it measurable.
Figure 4-11: Separation region - steady flow (No separation present in steady flow)

Figure 4-12: Separation region - unsteady flow
4.2.4 Other Loss Sources

There is a non-trivial remainder term in every row of the loss delineation table 4.2. That term represents flow phenomena not otherwise described or mentioned only in passing, such as profile loss, or the tip clearance loss.

Tip Clearance Loss

The largest single loss source left unaccounted for in the remainder category from table 4.2 is the tip clearance loss. However, investigation reveals that unsteady flow has minimal impact on the overall impact of this loss source. Figure 4-13 shows contours of non-dimensional $S''''_{visc}$ in steady flow, while figure 4-14 presents the same scenario with unsteady flow.

![Steady Simulation](image)

Figure 4-13: Steady tip clearance loss contour
4.3 Summary

This chapter quantified and assessed the loss generation of the full sector, and compared it to the results from the bladeless annular case for both steady and unsteady flow situations. The specific findings are delineated below.

1. Unsteady flow generates approximately 8% more loss than steady flow on average

2. Viscous shear layer formed by purge flow interaction with main flow is the primary source of the additional unsteady flow loss

3. Small plausible separation region at the corner of the hub and rotor pressure side at approximately 50% chord is also a source of increased loss under unsteady flow conditions
Chapter 5

Mechanism for Increased Loss Generation in the Shear Layer Due to Purge-Main Flow Interaction

Chapter 4 showed that the dominant loss source driven by purge flow injection is the viscous shear layer, while the secondary flow associated with the passage vortex system is not of a comparable level. Furthermore, some of the loss attributed to secondary flow was also suggested to be viscous shear layer induced. However, the exact source of increased loss in the unsteady viscous shear layer compared to steady viscous shear layer has not been established, and this topic will form the focus of this chapter. Unsteady flow brings two primary modifications to the main flow - purge flow interaction shear layer. These changes are inherent to the transition from a simplified steady state model to a full unsteady flow. The first is unsteady NGV-rotor interaction. The second is circumferential variation in the NGV pressure field, (which is mixed out circumferentially in the mixing plane approximation) that results in circumferential variations in the purge flow injection downstream of the NGV. This chapter will show that it is the latter flow feature that is the source of the additional loss identified in the previous chapter. This will be shown by isolating the effects of the two phenomena through tailored versions of the models described in Chapter 2.
5.1 Unsteady NGV-Rotor Interaction

The focus of the following section is on determining the importance of flow unsteadiness - primarily the NGV-rotor unsteady interaction, on loss generation. Two flow models will be employed to isolate unsteady NGV-rotor effects from those of circumferential variation. First, the 2D bladeless annular case will be examined, which due to its axisymmetric nature is incapable of responding to or creating circumferential variations. Second, computed results from a variant of the 3D bladeless annular model specifically tailored for isolating unsteady NGV-rotor interaction effects from circumferential non-uniformity effects will be interrogated. Ultimately, it will be shown that unsteady NGV-rotor interaction has no measurable impact on loss generation in the models used, and by extension assert that this phenomenon is responsible for little, if any, loss generated in the main flow - purge flow interaction in the hypothetical turbine stage.

5.1.1 2D Axisymmetric Model

The 2D axisymmetric case makes an efficient assessment of the impact of unsteadiness on purge flow. It models an axisymmetric flow in an annulus, with a constant inlet pressure to reflect the presence of the stationary NGV and sinusoidally oscillating back pressure to model the effects of the moving rotor pressure field. Circumferential variation, however, is completely excluded and thus the absence of any circumferential non-uniformities.

The results from this model under a variety of mass flow and swirl conditions are depicted in figures 5-1 and 5-2 respectively, which plot the axial variation in loss profiles measured via viscous lost work, normalized by the steady flow with 0% purge mass flow. This flow shows many of the same trends observed in Chapter 4: approximately linearly increasing loss with purge mass flow rate and non-linear reduction in loss with swirl added to injected purge flow. However, the computed results show no difference in loss level between steady and unsteady flow situations corresponding to imposed steady and unsteady back pressure. This is a valuable result because it involves a model that captures the heart of unsteadiness - unsteady rotor/stator impact on purge flow, while excluding purge flow circumferential variations, and produces effectively the same loss profile as the steady sim-
Cumulative $W_{\text{visc}}$ Losses through Annulus - Varying mass flow

Figure 5-1: 2D axisymmetric flow model: steady vs. unsteady cumulative viscous lost work comparison for varying mass. The region between the black vertical lines is the location of the purge slot.

ulation. The result is a compelling argument for the assertion that unsteady NGV-rotor interaction has little impact on loss generation but rather some other effect is responsible, such as the NGV imposed circumferential variation in pressure.

5.1.2 3D Bladeless Annular Model

The same result shown in the 2D axisymmetric flow model, namely that steady and unsteady flows have identical loss profiles, can be inferred from the computed results of figure 5-3 for the 3D bladeless annular flow model. The inlet boundary condition takes data
Figure 5-2: 2D axisymmetric flow model: steady vs. unsteady cumulative viscous lost work for varying swirl. The region between the black vertical lines is the location of the purge slot.

from the steady full sector case just after the mixing plane, maintaining the circumferentially uniform conditions seen by the steady state purge flow. The back pressure, however, is non-uniform circumferentially, modeling the presence of a rotor blade. Furthermore, it moves circumferentially at the same speed as the actual rotor, and has an impact similar to having an actual rotor just downstream of the end of the computational domain. Thus in contrast to the 2D axisymmetric situation, in the 3D bladeless annular model circumferential flow variations and unsteady flow induced by the moving rotor are permitted while circumferential flow variations in the inlet boundary condition are excluded. Due to difficulties in getting this somewhat elaborate case to converge under unsteady simulation,
fewer results are available, but they show the same trend shown in the previous section - unsteady NGV-rotor interaction has little apparent impact on time-average loss generation.

![Bladeless Annular Symmetric Model - steady vs. unsteady](image)

Figure 5-3: Bladeless annular flow model: steady vs. unsteady cumulative viscous lost work. The region between the black vertical lines is the location of the purge slot.

### 5.2 Circumferential Variations in Purge Flow

Unlike the impact of flow unsteadiness on loss generation induced by purge flow - main flow interaction, published literature on loss generation induced by circumferentially varying purge flow appears to be nonexistent. This section describes the use of two flow models directed at separating the loss generation driven by circumferential variations from the loss generation driven by unsteady NGV-rotor interactions. The first model is based upon the 3D bladeless annular configuration, but assesses the circumferentially varying boundary conditions using only steady simulation. The second will use the full sector configuration - but with computations implemented under frozen rotor approximation, locking the rotor
in place relative to the NGV. In contrast to the steady flow simulation based on the mixing plane approximation the mixing plane is removed, allowing for circumferential non-uniformities to propagate downstream. As will be presented in what follows, the computed results show that loss estimates accounting for the effects of imposed circumferential variations are nearly the same as the time average loss from the full sector computed unsteady flow.

5.2.1 3D Bladeless Annular Model

To assess the impact of circumferential variations on loss generation associated with the shear layer, a specialized configuration of the the 3D bladeless annular model is used. It is configured with the circumferentially varying inlet flow seen in the unsteady full sector simulation, but otherwise implemented as a steady flow computation. The result is a flow with a circumferential varying purge flow, but lacking a circumferentially translating rotor pressure field. The computed results from this flow configuration are assessed against the configuration lacking both the circumferentially translating outlet pressure field and the circumferentially varying inlet pressure field. This comparison isolates the effect of the NGV reflected in the pitchwise nonuniform inlet flow. The loss profiles from this comparison are depicted in figure 5-4. The purge mass flow rate is always 2.0% of the main flow, which is identical in all cases here.

As noted in section 4.2.2, the inlet condition for the bladeless annular model is taken after the mixing plane so that the bladeless annular model flow with pitchwise uniform inlet flow has incurred a mixed-out loss. Accounting for this, analysis shows that there is a measurable difference in loss between the circumferentially uniform and circumferentially varying purge flows in the bladeless annular model, even under steady flow conditions. Comparing this result to the result in figure 5-3 for the bladeless annular flow subjected to steady and unsteady conditions shows that it is the circumferential variations in purge flow that leads to additional loss generation, not the consequence of unsteady NGV-rotor interactions.
Figure 5-4: Bladeless annular cumulative viscous lost work computed for steady symmetric flow to compare with those computed for steady asymmetric flow for 0 swirl and 100% swirl. The region between the black vertical lines is the location of the purge slot.

5.2.2 Frozen Rotor Model

The second model used to establish that it is the circumferential variations in purge flow that results in increased loss generation is the frozen rotor model. This model is a modification on the full sector that was introduced in section 2.1.1; specifically it employs a steady flow configuration, in which the mixing plane between the rotor and stator domains has been removed. This preserves the NGV-induced circumferentially varying pressure field while eliminating unsteady NGV-rotor interactions. The frozen rotor computed results are assessed against the steady and unsteady flow configurations, and the assessments are elucidated in figure 5-5, which compares loss profiles between the steady, unsteady, and frozen rotor models. The cumulative loss generated in the unsteady and frozen rotor models is almost identical, with a small divergence beginning near the end of the rotor. The magnitude of this difference is small, arguably below the fidelity of the mesh, and does not
alter the conclusion. The frozen rotor model was responsible for at least as much loss as the full sector unsteady flow model, again demonstrating the significance of circumferential variations in purge flow on loss generation.

![Full Sector Model - Frozen Rotor Comparison](image)

Figure 5-5: Full sector comparison of cumulative lost work computed from unsteady (NGV-rotor stage) simulation, frozen rotor calculation and steady flow computation using mixing plane approximation. The region between the black vertical lines is the location of the purge slot.

### 5.3 Summary

The results described and presented in this chapter show that in unsteady flow, it is the circumferential variation in the purge flow ejection, not unsteady NGV-rotor interaction, that is predominantly responsible for the additional loss. The effect was isolated through:

1. 2D axisymmetric model that incorporates unsteady NGV-rotor interaction effects only
2. 3D bladeless annular model that incorporates unsteady effects but not circumferential variation

3. 3D bladeless annular model that incorporates effects due to circumferential variation but not flow unsteadiness

4. Frozen Rotor configuration of the full sector model allowing circumferential varying flow but no unsteady NGV-rotor interaction
Chapter 6

Two Stream Mixing and Parametric Analysis

In chapter 4 a straightforward loss comparison was presented between results from steady state flow and unsteady flow. It concluded that the bulk of the loss increase seen in unsteady flow was a result of mixing between main flow and purge flow, and occurred in the viscous shear layer. Chapter 5 determined that the source of this additional loss is induced by the circumferential variation in purge flow and not unsteady NGV-rotor interaction. This chapter analyzes and quantifies the loss associated with the viscous shear layer generated from a circumferentially varying purge flow interacting with the main flow.

Fundamentally, the viscous shear layer is simply the mixing region that adjoins the purge flow and near-hub main flow, and so an approach is defined here that quantifies the attendant loss. This approach is a control volume based model based on the work by Young and Wilcock, first mentioned in section 3.5.2 [15]. It will be shown that this model also forms the basis for implementing parametric assessment of loss generated from purge-main flow interaction that forms the second half of this chapter.

6.1 Mixing of Two Streams

The method for establishing the mixed out state of two streams used in this thesis was developed in response to the findings presented in Chapter 5. The procedure for implementing
the mixing-out analysis of two streams is similar to the work by Young and Wilcock [15]. The expression for the mixed out loss involving two streams is given below in equation 6.1:

\[
\frac{\dot{W}_{\text{loss}}}{m} \approx \frac{m_{st}}{m_{\text{total}}} \left( (v_{x,\text{main}} - v_{x,\text{purge}})^2 + (v_{r,\text{main}} - v_{r,\text{purge}})^2 + (v_{\theta,\text{main}} - v_{\theta,\text{purge}})^2 \right) 
\]

(6.1)

The key modification made in this equation is the mass scaling term, \(\frac{m_{st}}{m_{\text{total}}}\). The modification was implemented to allow the purge and main flows to be broken up into a number of streamtubes, which are weighted according to their relative mass flow when computing the total estimated loss for the mixing of the entire purge and main flows. Computational results showed that the mixing out of the purge flow and main flow principally involved mixing out of radial non-uniformity, hence the use of streamtubes with circumferential boundaries for mixing of main and purge flow. Furthermore, the mixed out state observed in CFD at the end of the computational domain is relatively uniform axially and radially, but maintains significant non-uniformities circumferentially. This effect is captured by the use of mixing out only of the individual streamtubes, rather than the fully mixed out state. \(^1\)

Figures 6-2 (bladeless annular) and 6-3 (full sector) show the regions from which velocity data from the main flow and purge flow was taken from CFD as inputs to the two stream mixing model. The locations of the streamtubes for main flow and purge flow are depicted in figure 6-1.

Examples of velocity profiles taken from these locations in the full sector are shown in figure 6-4. Notice that the steady flow velocity profiles are approximately uniform, whereas the unsteady flow velocity profiles vary substantially in comparison. This variation ties in to figure 6-5 which depicts the difference in the velocity components between main and purge flow, and shows that the circumferentially varying purge flow leads to circumferentially varying differences as well. The depicted differences constitute the inputs to equation 6.1. Note that the difference in net purge mass flow between the steady and unsteady flow varies by less than 1%.

The final result of the method is a number that estimates entropy generation from the

\(^1\)We use mixed-out state as distinct from fully mixed out state. The mixed out state would still retain a certain level of flow non-uniformity, principally the circumferential non-uniformity.
Figure 6-1: Schematic showing main flow, purge flow, and mixed flow around the purge slot. The locations of the main flow streamtube and the purge flow streamtube are identified.

mixing of the two input streams in isolation. By comparing steady against unsteady flow situations, or those with different injected purge mass flow, swirl, or purge slot sizing, the model allows one to estimate trends in loss from the mixing out of two streams.

6.1.1 Two Stream Mixing in the Bladeless Annular Model

The bladeless annular model is used to assess the applicability of Young and Wilcock's control volume for two stream mixing on the collection of streamtubes as described in the previous section. This is because the bladeless annular model was designed to isolate the mixing of the main and purge flows that forms the viscous shear layer. Results from the modified Young-Wilcock model will then be compared to viscous lost work computed from CFD to establish its adequacy in approximating relative changes in loss.
Figure 6-2: This figure depicts the locations from which main flow (red), purge flow (blue), and mixed flow (magenta) data was taken in the bladeless annular case to estimate the losses from mixing out of the shear layer associated with purge flow introduction.

**Comparison in Computed Viscous Lost Work from CFD and Control Volume Analysis**

Sample velocity profiles taken from the purge and main flow locations (depicted in figure 6-2) in the bladeless annular model are shown in figure 6-6. The figures show that the steady flow velocity profiles are approximately uniform, whereas the unsteady flow velocity profiles vary substantially by comparison. This variation ties in to figure 6-7 which depicts the difference in the velocity components between main and purge flow, and shows that the circumferentially varying purge flow leads to circumferentially varying differences as well. The depicted differences constitute the inputs to equation 6.1. Note that the circumferential velocity difference is significantly larger in magnitude than the difference in the other two velocity components and that velocity is made non-dimensional by the hub rotation speed (rotor rim speed). The circumferential velocity difference is the largest because the purge region is just after the stator swirls the flow in preparation of work extraction by the rotor, and as such the main flow's largest component is the circumferential velocity. In comparison, the purge flow is injected with 0 circumferential velocity and positive radial and axial velocities.
The parametric trend in loss generated, computed based on the bladeless annular model (for steady flow, unsteady flow, varying purge mass flow rate, varying purge flow swirl, etc), was determined based on control volume analysis, and assessed against computed results from CFD in table 6.1. The control volume results are in accord with those from CFD. This is expected, because the model only looks at the impact of circumferential flow variations on main flow - purge flow mixing. The value of this finding is that the control volume approach is useful and that the dominant loss generation arises from the velocity disparity between the purge and main flows.

6.1.2 Full Sector Mixing Model

The model given in equation 6.1 is applied to the full sector using two surfaces from which circumferential velocity profiles are taken: one for purge flow, the other for main flow.
Figure 6-4: Full sector, purge and main flow velocity profiles under steady and unsteady simulation. The top figures are taken from the main flow region denoted in figure 6-3, while the bottom figures are taken from the purge flow region.

These surfaces are depicted in figure 6-3. The result of applying the control volume analysis to the full sector for varying purge mass flow and swirl is presented in table 6.2.

Again the estimated loss ratios from control volume analysis and CFD are in accord despite the many other sources of loss generation computed in the full sector model. The implication is that the dominant source of loss is the shear layer arising from introducing the purge flow.

### 6.2 Parametric Analysis

One of the goals of the thesis, as stated in section 1.2, is to establish the relative importance of the various parameters characterizing purge flow injection, while quantifying their im-
Figure 6-5: Full sector, profile for difference in purge flow and main flow velocity components from steady and unsteady computations.

<table>
<thead>
<tr>
<th>Purge mass</th>
<th>Purge Swirl</th>
<th>Loss Ratio - CFD</th>
<th>Loss Ratio - Estimated (CV)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0</td>
<td>0</td>
<td>1.09</td>
<td>1.07</td>
</tr>
<tr>
<td>2.0</td>
<td>100</td>
<td>1.17</td>
<td>1.14</td>
</tr>
</tbody>
</table>

Loss Ratio is the ratio of loss generated in unsteady flow to that in steady flow.

The first two have been described and the latter two parameters are shown in Figure 6-8. The significance of each will be assessed for the unsteady flow situation.
Figure 6-6: Bladeless annular model, purge and main flow velocity profiles under steady and unsteady simulation. The top figures are taken from the main flow region denoted in figure 6-2, while the bottom figures are taken from the purge flow region.

### 6.2.1 Purge Mass Flow Rate

In his steady state flow analysis, Zlatinov found a linear relationship between purge mass flow rate and resultant loss - specifically 8% additional loss for each 1% additional purge mass flow rate. In unsteady flow, this linear trend is maintained. The result of parametric analysis on purge mass flow rate in the full sector, for both steady and unsteady flow, is presented in table 6.3 and figure 6-9. While the slopes of the lines are not identical, they are close. The resulting trend is an estimated 9% additional loss for each 1% additional purge mass flow rate under steady flow, and 8% additional loss per 1% purge mass flow rate under unsteady flow. Note that the unsteady flow loss increases less steeply with additional purge flow. This correlates with the finding from Chapter 5, that the blocked purge flow region reduces in circumferential extent as more flow is injected, which in turn
Figure 6-7: Bladeless annular model, profile for difference in purge flow and main flow velocity components from steady and unsteady computations.

Table 6.2: **Full sector Model**: CFD vs. CV

<table>
<thead>
<tr>
<th>Purge mass</th>
<th>Purge Swirl</th>
<th>Loss Ratio - CFD</th>
<th>Loss Ratio - Estimated (CV)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>0</td>
<td>1.082</td>
<td>1.074</td>
</tr>
<tr>
<td>1.0</td>
<td>0</td>
<td>1.069</td>
<td>1.057</td>
</tr>
<tr>
<td>2.0</td>
<td>0</td>
<td>1.050</td>
<td>1.045</td>
</tr>
<tr>
<td>2.0</td>
<td>100</td>
<td>1.078</td>
<td>1.065</td>
</tr>
</tbody>
</table>

Loss Ratio is the ratio of loss generated in unsteady flow to that in steady flow.

reduces the loss generated. The net effect is that unsteady loss increases more gradually than steady loss with additional purge flow mass. Note that the data point used for the 0% purge mass flow case with the slot is fortuitously in line with the linear parametric trend, whereas the corresponding data point (shown as a green dot on the ordinate in figure 6-9) with the slot removed is significantly lower. This is not surprising as it was noted in Section 4.1.2 and in figure 4.3, the unsteady flow situation with the slot generates an additional loss (it is hypothesized that this could be a result of the circumferentially varying pressure field associated with NGV); this is not so in the mixing plane steady flow approximation and in the unsteady flow situation with the slot removed.
6.2.2 Purge Flow Swirl

Zlatinov found that swirling purge flow reduced the loss from injecting purge flow in all cases involving hub-injected purge flow. However, he did not quantify the relationship beyond noting that it was approximately quadratic. This thesis confirms that the result extends to unsteady flow, and furthermore established scaling for the reduction and quantified the extent of the loss reduction.

The computed loss from the steady and unsteady full sector model flows under differing purge swirl is presented in table 6.4 and figure 6-10. In the figure, the y-axis denotes normalized loss, while the x-axis denotes the swirl of the injected flow. 100 indicates rim
Figure 6-9: Variation of purge flow induced loss generation with injected purge mass flow rate from steady and unsteady computations; the green data point on the ordinate is from the full sector unsteady computation for zero purge flow with the slot removed.

speed, and 150 approximately matches the main flow circumferential velocity of the near hub main flow. The choice of quadratic fit is based on equation 6.1; as swirling the purge flow reduces the circumferential velocity deficit, and the loss has a quadratic dependence on the circumferential velocity deficit. With the flow swirled to 150% of the rim speed, which is close to the circumferential velocity of the main flow above the hub, the loss is at a minimum.

6.2.3 Purge Slot Width and Injection Angle

The final two parameters Zlatinov initially identified as potentially relevant for controlling loss induced by purge flow introduction are the width (or axial size) of the purge slot where it meets the main flow path and the angle at which it is injected. These two parameters are indicated and defined in figure 6-8. His conclusion was that the loss does not vary significantly with these two parameters. Assessment of these two parameters in the unsteady
flow situation reaffirms this conclusion. It should be noted that the width and angle parametric examination was conducted using only the bladeless annular case, which looks only at mixing between the main and purge flow.

Modification of the slot in the bladeless annular model was carried out to generate two versions of the purge slot: one in which the width is decreased by 20% and the other in which the purge duct angle is increased by 7°. These variations are shown in figures 6-11 and 6-12, respectively. The results are presented in table 6.5, with the loss normalized by the value corresponding to the unmodified geometry situation. These results are also in accord with estimates from the control volume analysis. The mass flow and swirl are constant for these cases at the purge inlet.

**Width Modification**

Widening the duct will reduce both the axial and radial velocity of the flow as it leaves the duct. There are two factors that lead to this change having minimal loss impact. First, as
shown in figure 6-7 the radial velocity of the purge flow typically exceeds the main flow, but
the axial velocity of the purge flow is itself exceeded by the main flow. By lowering purge
velocity in both cases, loss from radial mismatch is reduced, but loss from axial deficit is
increased, so the effects responsible for the loss generation from the wider duct at least
partially offset each other. Second, the velocity deficit from either component is lower than
the circumferential deficit, which is unchanged.

**Angle Modification**

Modifying the duct angle will exchange purge flow axial and radial velocity - the purge
flow velocity magnitude is effectively unchanged, but its breakdown into axial and radial
components is not. From this, it can be concluded that angling the duct to match the main
flow path as closely as possible is beneficial, which matches computed results. However,
similar to the width modification, this change impacts only the radial and axial velocity
deficits of the purge flow - main flow interaction, with the dominant circumferential veloc-
ity term left unchanged. Correspondingly, the gain is minimal in comparison to other loss
sources identified thus far. In conclusion, it is demonstrably beneficial to align the purge
duct with the main flow, but the cost for a mismatch is so minimal that mechanical design
6.3 Summary

Control volume analysis for two streams, one representing the purge flow and the other the main flow, have been implemented. The results show that the circumferentially varying purge flow lead to additional loss which is in accord with computed results. The implication is that the additional loss generation associated with relieving the mixing plane approximation is essentially due to the circumferentially varying purge flow driven by the
NGV-induced pressure field. The role played by the NGV-rotor flow unsteadiness, if any, is small.

The results for control volume analysis using computed flow as inputs also show that of all the key parameters characterizing the purge flow, the mass flow rate and the swirl have the dominant impact on loss generation. The effects of purge slot width and purge duct angle, on the other hand, are relatively small. These results, though obtained here based upon an unsteady flow model, are in accord with the findings of Zlatinov for steady flow with a mixing plane approximation.
Chapter 7

Strategies to Reduce Purge Flow Circumferential Variation

The significance of purge flow circumferential variation as a driver for loss generation was discussed and described in chapter 5. Specifically in chapter 5 it was shown that circumferential variation in purge flow is primarily a consequence of the NGV-induced circumferentially varying pressure field. This leads to circumferential variations in purge flow ejection, which increases mixing loss, as described in section 6.1 using the control volume analysis for mixing out of two streams. The additional loss is above that of circumferentially uniform purge flow with identical mass flow rate. An implication of the finding is that reduction in mixing loss may be achievable through reducing the level of purge flow circumferential variation. Contouring the hub endwall offers such an opportunity, and the principal rationale behind sizing such changes and the preliminary results from these geometric modifications form the focus of this chapter.

The chapter begins by examining two possible implementations of endwall contouring on the bladeless annular model, one on the rotor side of the purge duct, the other on the stator side. The modifications rely upon different strategies for reducing purge flow circumferential variation, and the results and limitations from each are discussed. Lastly, a methodology is presented for sizing and implementing endwall contouring on the full sector, though the results from its implementation are not presented here.

There are two main criteria for analyzing the effectiveness of the endwall contouring.
First, how effectively does the contouring reduce purge flow circumferential variation. In the same vein, to what extent does the estimated loss based on control volume analysis of the mixing out of two streams decrease? The second criteria is the impact of endwall contouring on overall loss generation incurred due to any induced flow changes.

7.1 Bladeless Annular Model Endwall Contouring

The goal behind contouring the endwall in all cases is to counteract the circumferentially varying purge flow that is a result of stator wake influence. To this end, two modifications have been proposed, one upstream of the purge duct (this will be referred to as stator endwall modification/contouring) and one downstream (this will be referred to as rotor endwall modification/contouring), each of which uses a different mechanism to reduce purge flow circumferential variation. However, before discussing the endwall contouring, the nature of the primary cause of purge flow circumferential variation, the stator pressure field, must be characterized in section 7.1.1. Following that, the sizing of the modification to the rotor endwall is described, and the results obtained from its application to the bladeless annular model presented, in the context of addressing the two criteria delineated above. Likewise, the stator endwall modification to the bladeless annular model is, first sized and the results are discussed in light of the two stated criteria.

7.1.1 Characteristics of NGV-Induced Circumferentially Varying Pressure

As was alluded to in Chapter 5 and 6, the NGV-induced circumferentially varying pressure field results in a corresponding circumferential variation of the purge flow. It has been elucidated how such a circumferential variation in the purge flow results in additional loss generation. Now it is of engineering import to characterize the NGV-induced pressure field as it would provide useful guidelines for endwall contouring. A useful approximation for the NGV-induced pressure field would be that it satisfies the Laplace equation [16]:

\[ \nabla^2 p = 0 \]
\[ \nabla^2 p = 0 \]  

(7.1)

The dominant term in the solution to equation 7.1 is:

\[ P = P_0 e^{-\lambda x} \]  

(7.2)

where \( \lambda \) is the NGV pitch. Plugging in for this known value leads to the relation in equation 7.3:

\[ \frac{P}{P_0} = e^{-28x} \]  

(7.3)

Alternatively, the computed axial variation in NGV-induced pressure field for a specific circumferential profile is shown in figure 7-1. The reference pressure \( (P_0) \) is defined as the pressure at the NGV trailing edge. A curve fit of the computed axial variation in pressure is equation 7.4.:

Curve Fit on Computed Results: \( \frac{P}{P_0} = e^{-25x} \)  

(7.4)

where the exponent factor of 25 is quite close to the value of \( \frac{2\pi}{4}, 28 \). Thus at one pitch downstream of the NGV trailing edge, the NGV-induced pressure field would have decayed to just under 1% of its value at the NGV trailing edge. This is equivalent to \( \sim 4\% \) of dynamic head based upon rotor tip speed.

### 7.1.2 Rotor Endwall Axisymmetric Modification Description and Sizing

The basic concept behind the rotor endwall modification as introduced in section 2.1.1 is to delay injection of the purge flow until the circumferential variation in NGV-induced pressure field has decayed sufficiently. Given that the difference is no greater than 2% (of value at NGV trailing edge, which is \( \sim 8\% \) of dynamic head based upon rotor tip speed) as shown in figure 7-2, and with the rate of decay defined by equations 7.3 and 7.4, it is not
Figure 7-1: Downstream axial decay of NGV-induced pressure field from full sector CFD computations and exponential regression fit.
Figure 7-2: Contour of NGV induced static pressure field next to hub endwall to illustrate its impact on purge flow.
difficult to establish the point at which the pressure has decayed to below the purge flow stagnation pressure. The modification to the rotor endwall should extend at least this far, and realistically further to give some axial space for purge flow ejection. This sizing was applied to generate the modified cross-section shown in figure 7-3.

![Standard Modified Rotor Endwall](image)

Figure 7-3: Circumferential view of rotor endwall modification to bladeless annular flow configuration on the right to compare with standard (i.e. baseline) configuration on the left

### 7.1.3 Results from Modified Rotor Endwall

The rotor endwall modification in the bladeless annular model led to significantly reduced purge flow circumferential variation; this is elucidated in figure 7-4 which shows the circumferential variation of velocity components for modified rotor endwall (on the right hand side) to contrast against those for the baseline configuration (on the left hand side as the unmodified bladeless annular model). In that figure, notice that the velocity, especially the circumferential component, is much more uniform in the modified case than the unmodified situation. Use of control volume analysis in estimating loss shows that modified configurations shown in figure 7-3b result in a 75% loss reduction. The computed lost work gave essentially the same result, revealing that circumferential variation driven losses were reduced by 83%. The success of this modification when applied to the simplified bladeless annular model suggests that it should further be assessed using the full sector model.
Figure 7-4: Effect of modifying (rotor) endwall on altering circumferential variation of purge flow velocity: the baseline (unmodified) situation is shown on the left while the data from the modified endwall model is shown on the right

7.1.4 Axisymmetric Bladeless Annular Stator Endwall Modification

The idea behind the stator endwall modification is to accelerate the main flow just before the purge slot so as to redistribute the static pressure to result in a relatively more circumferentially uniform purge flow distribution. While a more complex version of endwall contouring is proposed for implementation in the full sector, the modification employed in the bladeless annular model here is axisymmetric for simplicity. The design of the stator endwall modification was introduced in section 2.1.1, but a radial-axial cross-section of the implementation in the bladeless annular model is shown in figure 7-5. The height is the only geometric parameter relevant here, and is sized in the next section.

Stator Endwall Height Sizing

The height of the ramp, \( h \), is set by the desired change in main flow velocity, which is in turn set by the pressure drop necessary to prevent purge flow from being blocked. Using values for the speed of sound that can be obtained through CFD, the pressure-Mach number relation, equation 7.5, relates changes in the main flow pressure to the main flow velocity.

\[
\frac{P}{P_i} = (1 + \frac{\gamma - 1}{2} M^2)^\frac{\gamma}{\gamma-1} \tag{7.5}
\]
Modifying the endwall should have negligible impact on the main flow or purge flow stagnation pressure, thus equation 7.6 should hold. Terms with the subscript "mod" denote quantities pertaining to the modified endwall case.

\[ P_t = P_{t-mod} \quad (7.6) \]

Referencing figure 7-2, it is clear that a 2% decrease in static pressure (of the value at the NGV trailing edge, equivalent to \( \sim 8\% \) of the the dynamic head based upon the rotor tip speed) of the main flow is sufficient to allow all purge flow to emerge from the purge slot into the main flow. Starting with a modified pressure-Mach number relation (equation 7.7), and with algebraic manipulation, it can be rewritten as equation 7.8 to yield the required Mach number for the modified endwall case. This establishes a Mach number relationship between the original Mach number of the main flow and the necessary Mach number in the modified case. With known speed of sound from CFD, this can be translated to a velocity relationship.

\[
\frac{P}{P_{mod}} = 1.02 = \frac{(1 + \frac{\gamma - 1}{2}M^2)^{\frac{1}{\gamma - 1}}}{(1 + \frac{\gamma - 1}{2}M_{mod}^2)^{\frac{1}{\gamma - 1}}} \]

\[ M_{mod} = \sqrt{\frac{2 + (\gamma - 1)M^2}{(1.02)^{\frac{1}{\gamma}}} - \frac{2}{\gamma - 1}} \quad (7.8) \]
To establish the necessary ramp height for accelerating the flow to a specified velocity, the 2D continuity equation is applied to a streamtube 1 length scale in height in equation 7.9. In this case, the length scale (w) represents the circumferential extent of the blocked purge flow region. The continuity equation assumes that the influence from the endwall shaping will not meaningfully propagate radially past this distance.

\[ \rho v w = \rho_{mod} v_{mod} (w - h) \]  

(7.9)

Neglecting any changes in density, this equation simplifies to equation 7.10, which shows the relationship between the target velocity ratio, length scale, and resulting necessary ramp height.

\[ \frac{v}{v_{mod}} = 1 - \frac{h}{w} \]  

(7.10)

### 7.1.5 Results from Modified Stator Endwall Bladeless Annular Model

The result of the stator endwall modification on the bladeless annular model is inconclusive. The implementation of stator endwall contouring did successfully reduce circumferential variation, as shown by the velocity profiles in figure 7-6. In that figure, notice that the velocity, especially the circumferential component, is much more uniform in the modified case than the unmodified situation. Correspondingly, the loss estimated from the control volume analysis of two stream mixing decreased by 58-64% (from modifying stator endwall to redistribute the NGV-induced pressure field to mitigate the circumferential flow distribution). However, the ramp also led to a small but significant separation of the main flow, which increased the loss generated by the stator with modified endwall (figure 7-5b) relative to the unmodified model (figure 7-5a) by 86%. This separation is discussed in the next subsection. As described in section 7.2.2 below, modifications will be made to the proposed implementation in the full sector to compensate for this separation.
Figure 7-6: Effects of modified (stator) endwall on altering the circumferential variation of purge flow velocity: the baseline (unmodified) situation is shown on the left, while the modified endwall is shown on the right.

**Separation Due to Stator Endwall Modification**

Figure 7-7 depicts the bladeless annular model with the modified stator endwall. It shows contours of loss generation (specifically nondimensional $T^S_{visc}$), and vectors of velocity. In the top figure, no purge mass is injected and thus no shear layer or substantial loss region should form. The presence of the empty duct, shown to result in (small) additional loss in chapter 4, cannot account for the magnitude of loss generated. However the presence of a recirculation zone as indicated by the velocity vector plot would strongly suggest the occurrence of flow separation.

The bottom figure shows the same situation but with a purge mass flow rate of 2% of the main mass flow rate. As such, a viscous shear layer is expected, and the loss region shown in the figure may not necessarily suggest the presence of any flow separation. However, this loss region appears to coincide with where separation loss might be expected. Thus it is suggested that the absence of the benefit in loss reduction as implied by the control volume analysis in the computed flow is most likely due to a flow effect not included in the two stream control volume analysis. It is presently hypothesized that this effect is associated with a flow separation implied by the results in the top figure of figure 7-7.
Figure 7-7: Contour of loss generation and corresponding plot of velocity vectors for modified stator endwall in the bladeless annular model with zero (top figure) and 2% (bottom figure) purge mass flow rate.
7.2 Suggestions and Recommendations for Full Sector End-wall Shaping

This section puts forward suggestions and recommendations for a proposed full sector implementation, and any attendant issues that may arise. Analyzing the new models should follow the same method used in the bladeless annular model of first determining whether circumferential variation reduction was achieved, then the implications on loss generation.

7.2.1 Full Sector Rotor Endwall Contouring

The implementation of the rotor endwall modification to the full sector model is effectively identical to that seen in the bladeless annular model. And given the effectiveness of the rotor modification on the bladeless annular model, the rotor endwall modification appeared the more promising of the two proposed changes.

7.2.2 Full Sector Stator Non- Axisymmetric Endwall Contouring

The basic mechanism for reducing circumferential variation in purge flow through stator endwall contouring in the full sector is unchanged from the bladeless annular model. It still relies upon accelerating the flow to reduce its static pressure. However, in light of the results from the bladeless annular model, a number of refinements have been implemented, and this section focuses on the sizing of these new parameters. The primary difference between the bladeless annular and full sector implementations is that the circumferential extent of the ramp has been reduced to only cover the region in which the purge flow is blocked. As such, the proposed stator endwall modification to the full sector is not axisymmetric. This is a significant departure from the bladeless annular implementation, which used an axisymmetric modification. Making this change should reduce the loss generation associated with the hypothesized induced flow separation by reducing the proportion of the flow that encounters the ramp. The hypothesized separation loss should also be reduced by the lower steepness of the ramp in the full sector implementation - an allowance that was not implementable in the bladeless annular model due to domain size constraints.
The approximate shape of this modification was introduced in section 2.1.1, and first depicted in figure 2-6. A more complete picture of the modification is presented in figure 7-8, while the subsequent figure 7-9 highlights the sizing parameters involved.

![Diagram of the modification](image)

**Figure 7-8:** Schematic drawing of full sector proposed stator endwall contouring

**Circumferential Position of Ramp and Ramp Angle**

The ramp is circumferentially positioned (see figure 7-8) to be in alignment with the main flow streamwise direction upstream of the slot. Unsteady effects, most notably the moving rotor pressure field, do adjust the region of blocked purge flow slightly over the course of each rotor blade passing, and the ramp is incorporated to address the entire circumferential extent of this (purge flow) blockage under all conditions. This constraint also sets the necessary width of the ramp.

A consequence of limiting the stator endwall modification to form a ramp is that an
Figure 7-9: Stator endwall schematic emphasizing relevant sizing parameters
additional parameter must be defined and set. This parameter is designated the ramp angle, and indicates the respective magnitudes of the axial and circumferential velocity components, as shown in figure 7-9. The ramp angle is determined by the time averaged velocity direction (x and θ components) of the flow responsible for blocking the purge flow injection.

**Calculation for Expected Upstream Influence of Asymmetric Stator Modification**

The final component of the stator endwall modification is the axial sizing. There are two primary considerations that influence this sizing. First, lengthening the ramp for a given height reduces the necessary deflection angle from the stator hub, and thus the amount by which the incoming main flow is turned. Second, while the point of this modification is to reduce the pressure of the local NGV-driven pressure circumferential variation, the pressure field the contouring creates will also extend upstream and influence the NGV. This effect can be described quantitatively using the Laplace’s equation whose solution has the leading term given in equation 7.2, but with the NGV pitch length scale replaced by the circumferential extent of the ramp (w). Based on the analysis using the Laplace equation, a ramp designed by the procedure given in this chapter would generate a pressure field that would modify the pressure at the NGV trailing edge by less than 0.2%.

**7.3 Summary**

In this chapter two main designs were presented for reducing the circumferential variations in purge flow, one focused on modifying the rotor endwall axisymmetrically, the other on modifying the stator endwall asymmetrically. They were assessed using the bladeless annular model, and were shown to reduce circumferential variation in both cases. However, only the rotor endwall modification successfully reduced loss. Suggestions and recommendations are also prescribed for modifying the stator endwall and rotor endwall for the full sector model to mitigate circumferential variation in purge flow.
Chapter 8

Conclusion and Future Work

This chapter will first provide a summary of the research described in this thesis. This is then followed by reiterating the key findings first outlined in the introduction of this thesis. Lastly, a discussion of potential future work, including two specific opportunities, will be presented.

8.1 Summary

The overarching goal of the research presented in this thesis was to assess the role of flow unsteadiness induced by NGV-rotor interactions on loss generation associated with the introduction of purge flow into the turbine main flow path. The technical approach for accomplishing this goal was as follows. The framework for establishing and quantifying loss was formulated based upon the work by Zlatinov [1], with expansion to include unsteady effects and circumferential non-uniformities due to removal of the mixing plane. Thus three-dimensional flow computations, unsteady as well as steady, have been carried out to assess the role of vane-rotor stage induced flow unsteadiness on loss generation from purge flow interacting with main flow. Computational models of varying physical and geometry complexities that range from a simple bladeless annular configuration to vane-rotor stage configurations were used for identifying the specific flow features responsible for loss generation; these were also complemented by control volume analyses of simple two streams flow model. The results have been post processed and interrogated to establish the findings.
delineated in the next section on “Key Findings”. It was determined that the vane-induced circumferential flow non-uniformity is responsible for the additional loss above that computed based on a mixing plane steady flow approximation for identical purge mass flow rate. In response, endwall modifications were designed to mitigate this source of additional loss. Preliminary assessments of these endwall contouring/modifications appear to show their effectiveness in loss mitigation. A procedure for sizing appropriate endwall contouring/modifications in the vane-rotor stage was formulated.

### 8.1.1 Key Findings

The key findings are as follows.

1. NGV induced circumferentially non-uniform static pressure field (and to a much lesser extent rotor blade induced bow waves) significantly increases loss generation in the turbine flow path upon introduction of injected purge flow with and without swirl. A mixing plane upstream of the purge slot, which renders the pressure field imposed on the purge slot circumferentially uniform, leads to an underestimation of the loss associated with purge flow - main flow interaction.

2. Flow path modification such as endwall contouring that reduces or eliminates NGV-induced static pressure circumferential non-uniformity imposed upon the purge flow has been shown to reduce circumferentially varying purge flow driven loss by ~ 75% or more. Example: if baseline loss (computed using mixing plane approximation) is 1.0, and loss computed with mixing plane removed is 1.08, flow path modifications can reduce it to 1.02.

3. Flow unsteadiness introduced by NGV-rotor interaction has marginal, if any, impact on loss generation from the introduction of purge flow into the main flow path. This is reflected in the frozen rotor simulation, which has a nearly identical loss profile as the unsteady simulation; the frozen rotor computations encompasses circumferentially varying purge flow driven loss but excludes unsteady flow driven loss.
4. Loss increases linearly with additional purge mass flow under both steady and unsteady conditions. Steady state analysis showed that a 1% additional purge mass flow leads to approximately 8.5% additional loss. In unsteady flow analysis 1% additional purge mass flow leads to approximately 7.5% additional loss. The difference in loss per mass flow occurs because the circumferential non-uniformity (primary new phenomena between the two situations) is sensitive to the quantity of injected mass flow and has proportionally greater effect with smaller purge mass flow rate. With purge flow circumferential velocity matching the main flow, the loss associated with that purge flow can be decreased by up to 85% relative to the situation of purge flow with no swirl. The purge flow induced loss has a quadratic dependence on the purge flow swirl (As the main flow swirl remains constant as the purge flow swirl is varied).

8.2 Future Work

This thesis concludes by putting forward useful future work that would extend and complement the findings of this thesis. The first major item described here, endwall contouring as a means to mitigate loss generated from purge flow interaction with main flow, is new based upon findings made here. The second, shroud injected purge flow, has not been addressed in this thesis though it has been assessed using steady flow mixing plane approximation by Zlatinov [1].

Endwall Contouring

The most immediate research task is to assess the performance benefits of the endwall contouring defined in section 7.2 in a full sector model. The proposed endwall designs are developed to mitigate the vane-induced circumferentially varying pressure field. The general viability of these changes and the argument for incorporation into the full sector is laid out in section 7.2. For assessing the use of endwall contouring in axial turbines with unsteady flow, 3 main metrics would be important. First, does the design reduce purge flow circumferential non-uniformity as intended? Preliminary assessments of their use in the bladeless annular model did so, and have been shown to be generally representative of full
sector behavior. Second, is net loss reduced through the reduction in purge flow circumferential variation? Again, the bladeless annular case suggests it should be. Lastly, how is the work output impacted by the modification? This is the one question that the bladeless annular model is incapable of addressing, but is needed for establishing the turbine stage performance impact of the changes induced by endwall contouring.

**Shroud-Injected Purge Flow**

The role of flow unsteadiness and vane induced circumferentially varying pressure field on loss generation associated with shroud-injected purge flow was not addressed in this thesis; there is thus a need to address this. Zlatinov found that increasing injected purge mass flow rate resulted in a tip clearance flow suppression with negligible net change in loss generation. In light of the mesh requirements alluded to in chapter 3 there is thus a need to revisit the results of Zlatinov on shroud-injected purge flow under steady flow approximation followed by assessing the role of vane induced circumferential variation in static pressure and flow unsteadiness.

The specific finding in this thesis on the role of vane-induced circumferential variation in static pressure field should apply to the situation of shroud injected purge flow in a vane-rotor stage unsteady flow environment with the removal of the mixing plane. The new consideration is rotor tip clearance flow, which is a source of loss. Zlatinov found that increasing purge mass flow rate resulted in tip clearance suppression, and effectively negligible loss contribution [1]. As noted earlier, Zlatinov used effectively the same mesh, and while steady simulation has lower requirements, review of his data reveals the some of the same issues that plagued the vane-rotor stage unsteady simulation incorporating the introduction of shroud injected purge flow. In light of this, the first step should be to revisit the steady flow computations including the mixing plane approximation but incorporating shroud injected purge flow. This is then to be followed by vane-rotor stage unsteady computations with shroud injected purge flow to assess the role of flow unsteadiness and the vane-induced circumferentially varying pressure field.
Bibliography


