Improved energy recovery through novel flow injection on a mixed flow turbine

Liu, Hao

2019

Liu, H. (2019). Improved energy recovery through novel flow injection on a mixed flow turbine. Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/137153>

<https://doi.org/10.32657/10356/137153>

This work is licensed under a Creative Commons Attribution-NonCommercial 4.0 International License (CC BY-NC 4.0).

Downloaded on 26 Apr 2023 18:15:17 SGT

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

SINGAPORE

**IMPROVED ENERGY RECOVERY THROUGH NOVEL
FLOW INJECTION ON A MIXED FLOW TURBINE**

LIU HAO

**Interdisciplinary Graduate School
Energy Research Institute @ NTU**

2019

**IMPROVED ENERGY RECOVERY THROUGH NOVEL
FLOW INJECTION ON A MIXED FLOW TURBINE**

LIU HAO

INTERDISCIPLINARY GRADUATE SCHOOL

A thesis submitted to the Nanyang Technological University in
partial fulfilment of the requirement for the degree of Doctor of
Philosophy

2019

Statement of Originality

I hereby certify that the work embodied in this thesis is the result of original research, is free of plagiarized materials, and has not been submitted for a higher degree to any other University or Institution.

19/08/2019

.....

Date

.....

Liu Hao

Supervisor Declaration Statement

I have reviewed the content and presentation style of this thesis and declare it is free of plagiarism and of sufficient grammatical clarity to be examined. To the best of my knowledge, the research and writing are those of the candidate except as acknowledged in the Author Attribution Statement. I confirm that the investigations were conducted in accord with the ethics policies and integrity standards of Nanyang Technological University and that the research data are presented honestly and without prejudice.

Input Date Here

19/08/2019

Date

Input Supervisor Signature Here

Alessandro Romagnoli

A handwritten signature in blue ink, consisting of stylized initials 'AR' followed by a long, sweeping horizontal stroke.

Authorship Attribution Statement

This thesis contains material from 3 papers published in the following peer-reviewed conference proceedings where I was the first and/or corresponding author.

Chapter 4 is published in part as:

- 1) H. Liu, A. Romagnoli, R. Martinez-Botas, S. Rajoo, and M. H. Padzillah, "Multi-Injection Turbine Housing: A Novel Concept for Tip-Leakage Improvement in Radial Turbines," in *ASME Turbo Expo 2017: Turbomachinery Technical Conference and Exposition*, Charlotte, NC, USA, 2017, pp. V02CT44A022-V02CT44A022: American Society of Mechanical Engineers.
- 2) H. Liu, A. Romagnoli, R. Martinez-Botas, S. Rajoo, and M. H. Padzillah, "Effects Of Secondary Flow Tip Injection On Performance And Losses In A Mixed Flow Turbine," in *2018 Global Power and Propulsion Forum*, Montreal, 2018: Global Power and Propulsion Society, 2018.
- 3) H. Liu, A. Romagnoli, M. I. Ismail, R. Martinez-Botas, S. Rajoo, and M. H. Padzillah, "Multi-Injection Turbine Housing: A Novel Concept For Performance Improvement In Radial Turbines," in *The 13th International Conference on Turbochargers and Turbocharging*, London, 2018: Institution of Mechanical Engineers.

The contributions of the co-authors are as follows:

- Asst/Prof Romagnoli and Prof Martinez-Botas provided the initial idea of the project and the direction of the research.
- I prepared the manuscript drafts. The manuscript was revised by Asst/Prof Romagnoli and Dr Srithar Rajoo.
- Dr Padzillah provided the baseline turbine data and geometry.
- Geometry preparation, meshing, CFD simulation of the turbine and analysis of the CFD results with injection was done by me.
- Dr Rajoo and Dr Padzillah assisted with the analysis of CFD results.
- Dr Ismail performed the 1-D engine simulations and helped to analyze the engine simulation results

19/08/2019

Date

Liu Hao

Abstract

Secondary flow injection is a way which allows for the efficiency of turbomachinery to be increased. In this thesis, a novel method for improving turbine performance using secondary flow injection through an injection slot over the turbine shroud is investigated. Numerical simulations were conducted on a mixed flow turbocharger turbine to test the effectiveness of secondary flow injection. Preliminary single passage simulations show that while injection was able to improve turbine blade loading and power by strengthening the tip leakage vortex, efficiency gains were minimal. The initial simulations also found that injection velocity and yaw angle were important parameters that affect turbine performance. An optimization was performed at the peak efficiency operating point at 50% turbine design speed to determine the injection set up which gives the highest turbine efficiency. The final optimized point resulted in an efficiency increase of 2.6 percentage points compared to the baseline scenario. Flow analysis shows that injection partially blocks the flow passage near the blade tip, forcing turbine passage flow to migrate towards the hub. This apparently weakens the hub suction side separation vortex and reduces entropy generation from the vortex. Using results from the optimization, a prototype was built. Experimental testing of the prototype was conducted and validated against full stage turbine CFD simulation results. Full stage turbine CFD results show that with inlet nozzle vanes, secondary flow injection did not result in any visible improvement in the internal flow field and entropy generation, but for an ideal injection overall efficiency can be improved by up to 2.28 percentage points at a velocity ratio of 0.75. Without nozzle vanes, however, secondary flow injection resulted in an efficiency improvement of up to 6.79 percentage points by weakening the hub suction side separation vortex and reducing its associated losses. Injection on the vaneless turbine configuration also resulted in a roughly 2 percentage point improvement in the peak turbine efficiency over the vaned turbine configuration. This might be due to more flow energy available to be extracted by the rotor from reduced losses due to the lack of nozzle vanes. Finally, a re-optimization of the injection was conducted by using the rotor inlet velocity profiles obtained from the full stage vaned turbine simulations as inlet boundary conditions on a single passage domain. Results from the re-optimization show that at VR higher than the peak efficiency point, the injected flow has reduced the entropy generation near the tip gap by disrupting the

formation of the tip leakage vortex. However, at VR near or below the peak efficiency point there is no clear efficiency improvement from flow injection. The maximum efficiency obtained was 2.71 percentage points. Overall, the research has shown that secondary flow injection has potential to increase turbine performance under some operating conditions.

Lay Summary

This thesis presents a novel concept in which air is injected over the casing of a small turbocharger turbine in order to increase its efficiency. Computer simulations of the flow within the turbine was used to investigate the effectiveness of the concept. Preliminary simulations using a simplified model of a single turbine blade passage show that while injection was able to increase turbine power output, efficiency gains were minimal because mixing and friction from injected flow caused additional loss in flow energy. The initial simulations also found that injection velocity and direction were important factors that affect the turbine performance. An optimization was performed at the peak efficiency operating point at 50% turbine design speed to determine the injection set up which gives the highest turbine efficiency. The final optimized point resulted in an efficiency increase of 2.6 percentage points compared to the baseline scenario. A detailed look at the flow within the turbine shows that injection forces the turbine flow to more closely follow the direction of the turbine blade, reducing losses arising from non-ideal flow. Using results from the optimization, a prototype was built. Experimental testing of the prototype was conducted and the results were used to validate against a full turbine simulation model. Simulations of the full turbine model show that for a turbine with nozzle vanes upstream, flow injection did not result in any changes in the flow that reduces losses, but an idealized injection can still improve efficiency by up to 2.28 percentage points. If there are no nozzle vanes upstream, flow injection can improve efficiency by up to 6.79 percentage points through improving the flow within the turbine and reducing losses. Injection on a vaneless turbine also resulted in roughly 2 percentage points higher maximum efficiency than injection on a vaned turbine. This might be because nozzle vanes introduce additional friction and reduce energy from the flow, and without them the turbine can extract more energy and produce more power. Finally, a second round of optimization was conducted for the injection using additional data obtained from simulations of the full turbine model. Results from the re-optimization show that at low turbine flow rates injection can reduce losses in the turbine, but at higher flow rates there is no clear efficiency improvement from flow injection. The maximum efficiency obtained was 2.71 percentage points. Overall, the research has shown that secondary

flow injection has potential to improve turbine performance under some operating conditions.

Acknowledgements

I would like to thank my supervisors, Assistant Professor Alessandro Romagnoli from NTU, and Professor Ricardo Martinez-Botas from Imperial College London, as well as my mentor, Professor Chan Siew Hwa, for their patient guidance and support on the project throughout the four years. Also, I would like to thank IGS and NTU for funding my education and giving me a chance to pursue research.

Special thanks go to Dr Srithar Rajoo for his technical advice on the experiments, and Dr Muhamad Hasbullah Bin Padzillah for his technical advice on the simulations, and for the thought provoking questions that helped open up new perspectives and ideas. My gratitude also goes to Mr Harminder Flora and Mr Asanka Munasinghe, who provided their technical expertise and helped tremendously with the experimental set up. I would also like to thank all my colleagues at Imperial College London, especially Peter, Uswah, Izzal, Bijie, Karl, Pepe, Karim and Vincent for your insights and advice regarding the simulations and experiments despite your own busy schedules. To the rest of the team: Esperanza, Eleni, Miles, Pablo, Alex, Xiaofeng and Tao, I thank all of you for a wonderful experience at Imperial College London.

To my colleagues in NTU: Jiaying, Yvonne, Antoni, Stefano, Dongxu, Lizhong, Chuan, Haoxin, Manuel, Alessio and Jun Onn, even though my project is quite isolated from the rest of you and we do not get to interact often, I am very grateful to have known all of you and appreciate all the help you have selflessly given over the course of my PhD. I am also very grateful for all the friends I have met in NTU throughout my 4 years of PhD here.

My gratitude also goes to Ms Huang Minying from ERI@N who has helped tremendously in dealing with administrative issues, as well as the other IGS staff for their support as well.

Last but not least, I give my thanks to Mom, Dad, my brother, as well as Kai Sing for always being there for me, letting me know that no matter what happens I always have them to rely on.

Table of Contents

Statement of Originality.....	v
Supervisor Declaration Statement.....	vii
Authorship Attribution Statement.....	ix
Abstract	xi
Lay Summary.....	xiii
Acknowledgements.....	xv
Table of Contents.....	xvii
Table Captions	xxi
Figure Captions	xxiii
Abbreviations	xxix
Chapter 1: Introduction	1
1.1 Introduction	2
1.2 Hypothesis/ Problem Statement	4
1.3 Objectives and Scope	4
1.4 Dissertation Overview.....	5
1.5 Findings and Outcomes/Originality	6
References	7
Chapter 2: Literature Review	9
2.1 Turbochargers and Turbocharging.....	10
2.2 Turbomachinery	13
2.2.1 Radial and Mixed Flow Turbines	15
2.3 Losses Mechanisms in Radial and Mixed Flow Turbines.....	19
2.3.1 Profile Loss	24
2.3.2 Tip Leakage Loss.....	28
2.3.3 Secondary/Endwall Loss.....	33
2.4 Methods to Reduce Turbomachinery Losses	37

2.4.1	Profile Loss	37
2.4.2	Tip Leakage Loss	37
2.4.3	Secondary/Endwall Loss	41
	References	44
	Chapter 3: Methodology	51
3.1	Overview of Multi-Injection Turbine Housing	52
3.2	Turbine Investigated	53
3.3	Simulation Methodology	55
3.3.1	Mesh Independence Study	56
3.3.2	Single Passage Simulations	57
3.3.3	Full Turbine Stage Simulations	61
3.4	Optimization Methodology	65
3.5	Design of Duct for Injection	69
3.6	Experimental Methodology	72
3.6.1	Mass Flow Measurement	74
3.6.2	Pressure Measurement	77
3.6.3	Temperature Measurement	77
3.6.4	Turbine Speed Measurement	78
3.6.5	Torque and Power Measurement	79
3.6.6	Uncertainty of Measurements	79
3.7	Performance Calculations	81
3.7.1	Mass Flow Parameter	81
3.7.2	Pressure Ratio	82
3.7.3	Isentropic Velocity Ratio	82
3.7.4	Isentropic Efficiency	82
3.7.5	Entropy Generation (CFD)	83
	References	85
	Chapter 4: Single Passage Simulations and Optimization	87
4.1	Baseline Simulations	88
4.1.1	Turbine Performance Curves	88
4.1.2	Detailed Flow Analysis	88

4.2	Initial Injection Simulations	94
4.2.1	Varying Yaw Angle	95
4.2.2	Varying Yaw Angle with Reduced Mass Flow	99
4.3	Optimization Results	102
4.3.1	Detailed Flow Analysis	104
	References	110
	Chapter 5: Full Stage Simulations and Validation With Experimental Results.....	111
5.1	Full Stage CFD Validation with Experiments.....	112
5.2	Effects of Injection in Full Stage Simulations	115
5.3	Potential Improvements Through Injection.....	115
	Chapter 6: Re-optimization of Injection Parameters	121
6.1	Investigation Procedure.....	122
6.2	Results	123
	Chapter 7: Conclusions and Future Work.....	131
7.1	Conclusions	132
7.2	Future Work	134
7.2.1	Engine Level Study	134
7.2.2	Unsteady Tests on Injection.....	134
7.2.3	Alternative Injection Designs	134
7.2.4	Injection on Vaneless Turbine	134

Table Captions

Table 3.1: Rotor A Geometric Parameters.....	54
Table 3.2: Single Passage CFD Boundary Conditions	59
Table 3.3: Injection Parameter and Ranges	60
Table 3.4: Turbine Operating Points, 30,000rpm	61
Table 3.5: Full Stage CFD Mesh Statistics	62
Table 3.6: Full Stage CFD Boundary Conditions.....	63
Table 3.7: Turbine Operating Points at 30,000rpm and 48,000rpm	64
Table 3.8: Description of Full Stage Injection CFD Cases.....	67
Table 3.9: Settings for Genetic Algorithm.....	69
Table 3.10: Uncertainty of measurements	80
Table 4.1: Summary of Injection Parameters Tested, Preliminary Investigation	95
Table 4.2: Final Optimized Parameters	103
Table 6.1: Single Passage Injection Re-Optimization Operating Points	122
Table 6.2: Re-optimization Injection Parameter Ranges	123
Table 6.3: Re-Optimization Results.....	124

Figure Captions

Figure 1.1: Global Temperature Anomaly, Relative to 1950-1980 Average [2].....	2
Figure 1.2: Comparison of Equivalent Petrol Car Emissions of Electric Vehicles in Various Countries [4].....	3
Figure 2.1: Typical Turbocharger Schematic [5].....	11
Figure 2.2: Typical performance characteristics: left) compressor right) radial turbine [4].....	11
Figure 2.3: Pulsating flow in a turbocharger turbine [6]	12
Figure 2.4: VGT Technologies [7]: Left) Sliding Nozzle, Right) Pivoting Vanes.....	13
Figure 2.5: Axial and Radial Turbine [11].....	14
Figure 2.6: Isentropic velocity ratio against flow inlet relative (β) and absolute angles (α) [14]	16
Figure 2.7: Mixed flow turbine schematic and characterization [14]	17
Figure 2.8: Typical axial, radial and mixed flow turbine efficiency against specific speed [6].....	18
Figure 2.9: Radial Turbine Loss Distributions: Left) Rohlik [29], Right) Huntsman & Hodson [30]	21
Figure 2.10: CFD Post-processed flow visualizations	22
Figure 2.11: Tip leakage loss on unshrouded blades [24]	29
Figure 2.12: Flow Over Blade Tip at: Left) 10% Meridional Chord, Middle) 50%, Right) 90% [78].....	30
Figure 2.13: Secondary Flow Structures in Axial Turbines: Left) Schematic [86], Right) LES Simulation [87]	34
Figure 2.14: Flow Near Leading Edge. Left and Middle: Passage Secondary Flow Direction and Vorticity Contours (Palfreyman & Martinez-Botas [16]), Right: Passage Limiting Streamlines (Chen et al. [90])	36
Figure 2.15: Limiting Streamlines on Leading Edge Hub and Blade Suction Surface, Chen et al.[90].....	36
Figure 2.16: Geometry Modifications: Left) Winglet and Squealer [94], Right) Shroud Endwall Contouring [98]	39
Figure 2.17: Left) Casing Injection, Behr et al. [101] Right) Honeycomb Tip Injection, Wang et al. [102].....	41

Figure 2.18: Reducing Endwall Losses: Left) LE Bulb (Sauer et al., 2001 [109]), Right) Enwall Contouring (Turgut & Camci, 2015[110])	42
Figure 3.1: Schematic of turbine geometry with secondary injection slot	52
Figure 3.2: Definition of Injection Parameters	53
Figure 3.3: Overall Research Methodology Adopted	53
Figure 3.4: 3D Model of Rotor A	54
Figure 3.5: Schematic of Volute [2]	55
Figure 3.6: Schematics of Nozzle Vane [2]	55
Figure 3.7: Mesh Independence Results	57
Figure 3.8: Secondary injection single passage domain	58
Figure 3.9: Single Passage CFD Mesh	58
Figure 3.10: Limits of Injection Location and Width	60
Figure 3.11: User-defined Surfaces for Turbine Inlet and Outlet Measurements.....	61
Figure 3.12: Full stage turbine CFD simulation domain with injection: left) with injection duct, right) without injection duct.....	62
Figure 3.13: Full Stage User-defined Surfaces for Turbine Inlet and Outlet Measurements	64
Figure 3.14: Full Stage Injection CFD Cases	66
Figure 3.15: Crossover and Mutation in Genetic Algorithm	67
Figure 3.16: General workflow for the current optimization process.....	68
Figure 3.17: Area and radius distribution of designed injection volute.....	71
Figure 3.18: Final design of injection volute	71
Figure 3.19: Manufactured injection duct.....	72
Figure 3.20: Schematic of turbocharger test rig [8]	73
Figure 3.21: Experimental set up with injection	74
Figure 3.22: Schematics of the V-Cone Flow Meter	75
Figure 3.23: Schematic of Nozzle Flow Meter	76
Figure 3.24: Turbine Speed Measurement [11]	79
Figure 4.1: Performance Curves of Baseline Turbine, Single Passage CFD.....	89
Figure 4.2: Streamlines within passage for baseline, suction side. Top: Point 1, Middle: Point 5, Bottom: Point 8.....	90
Figure 4.3: Streamlines within passage for baseline, pressure side, Point 1	91
Figure 4.4: Surface streamlines for baseline, suction side, Point 8	92

Figure 4.5: Contours of entropy generation rate for baseline, suction side. Top: Point 1, Middle: Point 5, Bottom: Point 8.....	93
Figure 4.6: Contours of entropy generation rate for baseline, pressure side, Point 1 ..	94
Figure 4.7: Change in turbine power due to injection over baseline scenario	96
Figure 4.8: Blade loading at 95% span for injection at different yaw angles compared with baseline, VR=0.7	97
Figure 4.9: Tip leakage vortex pressure contours and streamlines at VR= 0.7 for: a)baseline, b)injection 0° yaw, c)injection 30° yaw, d)injection 45° yaw	98
Figure 4.10: Change in turbine efficiency due to injection over baseline scenario	99
Figure 4.11: Change in turbine power compared to baseline scenario for reduced mass flow case	100
Figure 4.12: Blade loading at 95% span for injection with reduced mass flow at different yaw angles, compared with baseline, VR=0.7	101
Figure 4.13: Tip leakage vortex pressure contours and streamlines at VR= 0.7 for: a)baseline, b)injection 0° yaw, c)injection 30° yaw (mass flow reduced), d)injection 45° yaw (mass flow reduced)	101
Figure 4.14: Change in efficiency compared to baseline scenario for reduced mass flow case.....	102
Figure 4.15: Turbine efficiency curve for baseline and injection cases	104
Figure 4.16: Turbine Pressure Ratio VS MFP Curve for Baseline and Injection Cases	104
Figure 4.17: Streamlines within passage for injection, suction side. Top: Point 1, Middle: Point 5, Bottom: Point 8.....	106
Figure 4.18: Contours of entropy generation rate for injection, pressure side, Point 1	107
Figure 4.19: Contours of entropy generation rate for injection, suction side. Top: Point 1, Middle: Point 5, Bottom: Point 8.....	108
Figure 4.20: Tip leakage Flow Structure, Point 5. Left: Baseline, Right: Injection ..	109
Figure 5.1: 30,000rpm CFD validation results: Top) Pressure Ratio (PR) against Mass Flow Parameter (MFP) Middle) Torque against Velocity Ratio (VR) Bottom) Efficiency against Velocity Ratio (VR).....	113

Figure 5.2: 48,000rpm CFD validation results: Top) Pressure Ratio (PR) against Mass Flow Parameter (MFP) Middle) Torque against Velocity Ratio (VR) Bottom) Efficiency against Velocity Ratio (VR).....	114
Figure 5.3: Rotor Streamline Comparison: Left) Single Passage CFD, Right) Full Stage CFD.....	115
Figure 5.4: Efficiency Curves of Different Vaned Turbine Injection Cases Compared with Baseline, 30,000rpm	117
Figure 5.5: Velocity Streamlines for Vaned Turbine: left) Baseline Vaned Full Stage, Middle) Injection Case C, Right) Injection Case D	117
Figure 5.6: Normalized Entropy Generation Rate Contours for Vaned Turbine: left) Baseline Vaned Full Stage, Middle) Injection Case C, Right) Injection Case D	118
Figure 5.7: Efficiency Curves of Different Vaneless Turbine Injection Cases Compared with Baseline, 30,000rpm	119
Figure 5.8: Velocity Streamlines for Vaneless Turbine: left) Baseline Vaneless Full Stage, Middle) Injection Case E, Right) Injection Case F	119
Figure 5.9: Normalized Entropy Generation Rate Contours for Vaneless Turbine: left) Baseline Vaneless Full Stage, Middle) Injection Case E, Right) Injection Case F ...	119
Figure 6.1: Re-optimization Simulation Domain, With User-defined Turbine Inlet and Outlet Surfaces	123
Figure 6.2: Re-optimized Injection Efficiency Comparison Against Baseline: Top) 30,000rpm, Bottom) 42,000rpm	125
Figure 6.3: Entropy Generation Contours and Streamlines for Leakage (Yellow) and Injection (Red) Flow, Point 30-3: Left) Baseline, Right) Re-optimized Injection	126
Figure 6.4: Entropy Generation Contours and Streamlines for Leakage (Yellow) and Injection (Red) Flow, Point 30-5: Left) Baseline, Right) Re-optimized Injection	126
Figure 6.5: Entropy Generation Contours and Streamlines for Leakage (Yellow) and Injection (Red) Flow, Point 30-7: Left) Baseline, Right) Re-optimized Injection	127
Figure 6.6: Entropy Generation Contours and Streamlines for Leakage (Yellow) and Injection (Red) Flow, Point 42-5: Left) Baseline, Right) Re-optimized Injection	127
Figure 6.7: Entropy Generation Contours and Streamlines for Leakage (Yellow) and Injection (Red) Flow, Point 42-7: Left) Baseline, Right) Re-optimized Injection	128
Figure 6.8: Entropy Generation Contours and Streamlines for Leakage (Yellow) and Injection (Red) Flow, Point 42-9: Left) Baseline, Right) Re-optimized Injection	128

Abbreviations

<i>CFD</i>	Computational fluid dynamics
<i>EGR</i>	Exhaust gas recirculation
<i>ICE</i>	Internal combustion engine
<i>LE</i>	leading edge
<i>LES</i>	large eddy simulation
<i>MFP</i>	Mass flow parameter
<i>RANS</i>	Reynolds Averaged Navier-Stokes
<i>rpm</i>	Revolutions per minute
<i>PR</i>	Total-to-static pressure ratio
<i>PS</i>	Pressure surface
<i>SS</i>	Suction surface
<i>TE</i>	Trailing edge
<i>VGT</i>	Variable geometry turbocharger
<i>VR</i>	Isentropic velocity ratio

Chapter 1: Introduction

In order to reduce carbon emissions, the transport industry is moving towards increasing electrification. However, while energy production is still dominated by fossil fuels, improving internal combustion engine (ICE) performance will still play a key role in reducing emissions. The turbocharger is an important energy recovery device that can help improve the efficiency of ICEs. This thesis presents a novel method of secondary flow injection from the turbine casing to improve the rotor internal flow field and increase turbine efficiency. This chapter highlights the research problem, research objectives and the scope of the thesis, as well novel outcomes of the research. An overview of the entire thesis is also presented.

1.1 Introduction

Global warming is arguably one of the most serious problems facing humankind in recent decades. Global temperatures have risen by 0.8°C relative to 1951-1980 average temperatures, as can be seen in Figure 1.1. Under the Paris Agreement, countries are committed to taking action to limit global warming to well below 2°C above pre-industrial levels [1].

Source: climate.nasa.gov

Figure 1.1: Global Temperature Anomaly, Relative to 1950-1980 Average [2]

Globally, the transport industry is a major contributor to carbon dioxide emissions, contributing about 20% of the total carbon dioxide emissions [3]. Therefore, reducing carbon emissions from the transportation sector will be one of the keys to meeting the Paris Agreement objectives. For automotives, current methods of reducing emissions include: engine downsizing for internal combustion engines (ICEs), hybrid-electric vehicles (HEVs), fuel-cell electric vehicles (FEVs), as well as battery electric vehicles (BEVs). Although BEVs and FEVs have zero tailpipe carbon emissions, depending on how electricity is generated in the area, overall carbon emissions from these vehicles might not actually be lower than conventional ICEs or HEVs. It is estimated by Shrink That Footprint, an independent research group that EVs in places where power generation is mostly coal based, such as China and India, perform no better than petrol vehicles, as shown in Figure 1.2 [4], while in many countries with a mixed power generation profile such as the USA and UK, EV emissions are comparable to hybrid vehicles. These numbers will surely improve as more renewable energy sources enter

the energy mix, but in the meantime improving ICE performance and fuel consumption will continue to play a key role in reducing emissions.

Figure 1.2: Comparison of Equivalent Petrol Car Emissions of Electric Vehicles in Various Countries [4]

Turbochargers are the most widely used commercial automotive energy recovery systems. They are forced induction devices used to increase the engine's efficiency and power by recovering part of the exhaust energy and using it to increase the airflow entering the engine. Turbocharged engines produce greater power compared to a naturally aspirated one of the same displacement. Thus, turbochargers result in greater engine efficiency and allows for downsizing of engines which further reduce fuel consumption through weight reductions. In a vast and economically substantial automotive market, any increase from the currently rated efficiency of turbochargers will have extended list of advantages: lower fuel consumption, higher engine power, and meeting the ever more stringent emission standards (NEDC, UDC). Even as the industry moves towards increasing electrification, turbochargers will continue to serve

as a key enabler in emissions reduction and increased fuel efficiency. Garrett-Advancing Motion estimates an additional 24 million turbocharged cars will be added to the market by 2025 [5].

1.2 Hypothesis/ Problem Statement

The turbocharger turbine is the device responsible for recovering the exhaust gas energy from the engine. Thus, any improvements on the turbine efficiency will help improve the overall performance of the turbocharger.

There has been much research conducted to improve the performance of turbomachinery, but there are still significant gaps in the knowledge of turbomachine design and optimization, especially in radial turbomachines, which are commonly used in small-scale turbocharger applications for energy recovery. In this research we aim to address some of the gaps in radial turbomachinery research and advance passive/control systems to improve turbine performance by introducing novel concepts for the housing design. For this thesis, it is proposed that secondary flow injection over the shroud be used as a method of improving the flow field within a turbine rotor passage, thereby reducing losses and increasing turbine efficiency. It is hypothesized that injecting flow in a certain direction and speed will positively interact with the tip leakage as well as the passage flow and reduce tip leakage and passage flow losses.

1.3 Objectives and Scope

The objectives of this research are:

- 1) Evaluate the feasibility in using secondary flow injection from the turbine casing to reduce losses within a mixed flow turbocharger turbine and improve efficiency.
- 2) Gain greater understanding of the flow structures inside a mixed flow turbine and how they interact with flow injection.

The scope of the research will be mainly on the fluid mechanics in turbomachinery. Even though heat transfer effects will affect turbomachinery performance as well, studies and analysis of heat transfer will not be the main focus.

1.4 Dissertation Overview

The thesis describes and validates the design and optimization of a novel concept in which secondary flow is injection over the shroud of a small mixed flow turbine used in turbocharger applications to improve the flow field of the turbine and reduce aerodynamic losses, thereby improving the efficiency of the turbine.

Chapter 1 provides a rationale for the research and outlines the goals and scope.

Chapter 2 reviews the literature concerning the historical and contemporary developments in radial and mixed flow turbines research for use in automotive turbochargers. A large part of the review is dedicated to the research on loss mechanisms and mitigation techniques used in turbomachinery.

Chapter 3 discusses the methodology used in the investigation of the multi-injection turbine housing concept. Both the numerical and experimental methods are highlighted, as well as the optimization strategy adopted.

Chapter 4 discusses the results from the initial single passage simulations as well as the optimization results. The results are presented in the form of turbine performance parameters, as well as detailed flow structure analysis from the CFD results.

Chapter 5 discusses the experimental results, together with the full stage CFD validation of the experimental results. The chapter also explores the potential of the multi-injection turbine housing through additional full stage simulations for a vaneless turbine.

Chapter 6 elaborates on the strategy and results for a re-optimization of the injection parameters using data obtained from the full stage turbine simulations.

Chapter 7 summarizes the main findings of the research, as well as touch on potential future studies that is useful for furthering understanding and application of the multi-injection turbine housing.

1.5 Findings and Outcomes/Originality

This research led to several novel outcomes by:

1. Providing a detailed understanding of the flow structures within a mixed flow turbine at various off-design operating points.
2. Evaluating the feasibility of using secondary flow injection as a method of improving the efficiency of a mixed flow turbine used in automotive turbochargers.
3. Using optimization to find out the optimum injection parameters to be used.

References

- [1] United Nations Framework Convention on Climate Change. (2019). *The Paris Agreement* | UNFCCC. Available: <https://unfccc.int/process-and-meetings/the-paris-agreement/d2hhdC1pcy>
- [2] NASA. (2019). *Global Climate Change: Vital Signs of the Planet*. Available: <https://climate.nasa.gov/vital-signs/global-temperature/>
- [3] Our World in Data. (2019). *CO2 and Other Greenhouse Gas Emissions - Our World in Data*. Available: <https://ourworldindata.org/co2-and-other-greenhouse-gas-emissions#emissions-by-sector>
- [4] L. Wilson, "Shades of Green Full Report," Shrink That Footprint 2013, Available: <http://shrinkthatfootprint.com/wp-content/uploads/2013/02/Shades-of-Green-Full-Report.pdf>.
- [5] Garrett-Advancing Motion, "Garrett Motion Inc. Investor Conference Executive Overview," ed, 2018.

Chapter 2: Literature Review

This chapter reviews the literature on topics related to turbochargers, radial and mixed flow turbines, as well as turbine losses. A brief history of turbocharger development is given, followed by some current research trends in turbocharger research. Literature of research on radial and mixed flow turbines is also reviewed. This is followed by a review of the current state of the art in terms of understanding, predicting, and mitigating turbine losses, with a focus on radial and mixed flow turbines. The literature review showed gaps in the understanding of radial and mixed flow turbine aerodynamics, as well as relatively sparse investigations into methods to improve the aerodynamic efficiency of radial and mixed flow turbines, especially in terms of tip leakage and secondary flow.

2.1 Turbochargers and Turbocharging

The turbocharger was invented by the Swiss engineer Alfred Buechi, who first filed a patent in 1905 for a concept of improving internal combustion engine (ICE) performance by driving supercharger compressor using exhaust gas energy recovered using turbines [1]. Early usage of turbochargers were mainly in aircraft, marine, and heavy-duty applications [2], and eventually turbochargers made their way into smaller automotive engines. Today, due to the need for ever more efficient engines that minimize fuel consumption and lower CO₂ emissions, turbochargers have become a common feature in modern passenger vehicles. Globally, around 45% of light vehicles are turbocharged as of 2017, and Garrett Motion estimates turbo penetration to reach 61% by 2025 [3], with turbochargers being used in three quarters of hybrid vehicles by then.

A typical turbocharger consists of a compressor and a turbine joined by a common shaft, as shown in Figure 2.1. Exhaust gas from the engine passes through the turbine, which recovers part of the exhaust gas energy, converting it into torque. The torque then drives the turbocharger compressor, which compresses inlet air to a higher pressure before discharging it into the engine. The result is that air intake is increased by the turbocharger, which increases the engine power. Thus, a turbocharger recovers energy from the exhaust gas, which would otherwise be wasted, to increase the engine power output.

The automotive engine is designed to operate over a wide range of operating conditions. However, the turbocharger compressor and turbine are essentially steady-state devices, as turbomachinery performance is heavily dependent on the flow angle at the entry to the compressor impeller or the turbine rotor [4]. Away from the design point, compressors and turbines suffer a significant drop in efficiency. Figure 2.2 shows typical compressor and radial turbine performance characteristics. As can be seen from the figures, compressor and turbine efficiency (η) drop significantly away from the design point. To add to the problem, the exhaust gas from the ICE is highly pulsating in nature due to the reciprocating engine, as shown in Figure 2.3, which means that the turbocharger turbine operates under highly unsteady flow conditions.

Figure 2.1: Typical Turbocharger Schematic [5]

Figure 2.2: Typical performance characteristics: left) compressor right) radial turbine [4]

Figure 2.3: Pulsating flow in a turbocharger turbine [6]

Much work has been conducted to improve the matching between turbocharger and the internal combustion engine (ICE). One of the most prominent methods of improving turbocharger performance under varying operating conditions is the variable geometry turbocharger (VGT). Two common types of VGT systems currently used are the sliding nozzle vanes and the pivoting nozzle vanes [7] used in turbocharger turbines, shown in . Both systems aim to improve off-design turbine performance by varying the aspect ratio of the volute to control the inlet speed to the turbine rotor. Pesiridis & Martinez-Botas [8] took the VGT concept a step further and introduced a novel active flow control method that optimizes the inlet area of a turbocharger turbine with respect to the incoming pulsating exhaust flow to take advantage of the pulsatile energy that fluctuates over a cycle. Their design made use of a sliding wall, similar to the sliding nozzle vanes, to control the turbine inlet area according to the pulsating flow of the exhaust. Their design was proved to be advantageous over the standard VGT model. Rajoo [9] explored an active control method for pulsating flows using pivoting nozzles. The results show that by allowing the nozzles to oscillate naturally against a spring pre-load, better energy extraction could be achieved at lower pulse frequencies (which correspond to lower engine speeds). More recently, Cao [10] studied a passive rotating nozzle vane design, and found that by allowing the nozzle ring to rotate freely to the pulsating flow, the inlet flow to the turbine rotor can be regulated, resulting in higher minimum and cycle-averaged efficiency. Cao also explored actively controlling the rotation speed of the nozzle, but found no obvious performance benefits. Variable geometry devices were also explored in turbocharger compressors to prevent choke or surge, and these were covered in a review by Feneley et al. [7], and will not be covered

in detail in this review. Despite the benefits of VGTs in improving the operating range of turbocharger turbines, there are certain drawbacks as well. The most obvious drawback is the additional production and maintenance cost of VGTs due to the additional moving parts. Durability might also be an issue [7].

Figure 2.4: VGT Technologies [7]: Left) Sliding Nozzle, Right) Pivoting Vanes

2.2 Turbomachinery

The most important components of a turbocharger are the turbomachinery: the compressor and the turbine. Turbomachinery are devices which transfer energy between a working fluid and a rotor. Turbomachinery have been used for a wide variety of applications, ranging from pumps, aircraft engines, to turbochargers in both stationary and automobile applications. There are two main types of turbomachinery: the axial-flow type, and the centrifugal/radial type. Axial-flow turbomachinery has flow travelling along the axis of rotation, while the flow in radial/centrifugal turbomachinery undergoes a 90° turn through the radial direction. Figure 2.5 shows a typical axial and radial turbine schematic.

Figure 2.5: Axial and Radial Turbine [11]

Axial flow turbomachinery are mainly used in aircraft jet engines, while the radial/centrifugal type are the main type used in automotive turbochargers. The reason for using radial/centrifugal type turbomachinery in automotive turbocharger applications is mainly due to the higher specific power of radial-flow type machines [6]. This makes it an attractive choice for a single stage system that is lightweight and cost-efficient.

The work done/work extracted by a compressor/turbine can be derived starting from the tangential momentum equation of the fluid [11], which results in a net torque:

$$\tau = \dot{m}(r_2 C_{\theta 2} - r_1 C_{\theta 1}) \quad (2-1)$$

The power of the rotor can then be obtained by multiplying the torque by the angular velocity of the rotor:

$$\dot{W} = \omega \tau = \dot{m} \omega (r_2 C_{\theta 2} - r_1 C_{\theta 1}) \quad (2-2)$$

The specific power of the rotor is then:

$$\frac{\dot{W}}{\dot{m}} = U_2 C_{\theta 2} - U_1 C_{\theta 1} \quad (2-3)$$

The above equations are commonly known as the Euler Turbomachinery Equations, and are equally applicable to both turbines and compressors. For turbines, station 2 is the turbine inlet and station 1 is the turbine outlet, while for compressors the reverse is true. The specific power of the rotor can also be expressed as:

$$\frac{\dot{W}}{\dot{m}} = U_2 C_{\theta 2} - U_1 C_{\theta 1} = h_{02} - h_{01} = h_2 - h_1 + \frac{1}{2}(C_2^2 - C_1^2) \quad (2-4)$$

Assuming an adiabatic, irreversible flow process, the flow rothalpy is constant:

$$I = h_{01,rel} - \frac{1}{2}U_1^2 = h_{02,rel} - \frac{1}{2}U_2^2 \quad (2-5)$$

$$h_{02,rel} - h_{01,rel} = \frac{1}{2}(U_2^2 - U_1^2) \quad (2-6)$$

Since $h = h_{0,rel} - \frac{1}{2}w^2$:

$$\begin{aligned} \frac{\dot{W}}{\dot{m}} &= h_2 - h_1 + \frac{1}{2}(C_2^2 - C_1^2) \\ &= h_{02,rel} - h_{01,rel} + \frac{1}{2}(w_2^2 - w_1^2) + \frac{1}{2}(C_2^2 - C_1^2) \\ &= \frac{1}{2}(U_2^2 - U_1^2) + \frac{1}{2}(w_2^2 - w_1^2) + \frac{1}{2}(C_2^2 - C_1^2) \end{aligned} \quad (2-7)$$

The above equation [12] expresses the work transfer of a turbomachinery in terms of the fluid and rotor velocities. For an axial turbomachinery, $U_2 = U_1$ as there is no change in radius, while for a centrifugal compressor/radial inflow turbine $U_2 > U_1$ thus this term is positive. This results in a higher specific work by a radial turbomachine compared to its axial counterpart.

2.2.1 Radial and Mixed Flow Turbines

Today, turbines used in automotive turbocharger applications are almost entirely of the radial flow type. Besides the turbocharger market, radial turbines are also used in some air-conditioning and refrigeration units [6]. They are also a common option used in organic rankine cycle systems, usually for higher pressure ratios and lower flow rates [13]. The term radial turbine refers to both radial inflow and radial outflow turbines, but in this thesis it will be used to refer to radial inflow turbines only.

As previously mentioned, a radial turbine has higher specific power output compared to its axial counterpart. In addition, automotive turbochargers are typically small in size,

and at this size axial turbine blades have very short blade heights and narrow flow passages, resulting in larger boundary layer blockage [4]. For small sizes, radial turbines are also cheaper to manufacture [4, 6]. This makes radial turbines a more appropriate choice for automotive turbochargers. However, radial turbines have their drawbacks as well. The main drawback of the radial turbine is that, due to strength constraints [4, 6], radial turbine blades are radially fibred, which constrains the inlet blade angle angle to 90° . As a result, radial turbine efficiency peaks at an isentropic velocity ratio of 0.7. This can be seen in Figure 2.6 below, which shows the relationship between peak efficiency velocity ratio versus the flow absolute and relative angles.

Figure 2.6: Isentropic velocity ratio against flow inlet relative (β) and absolute angles (α) [14]

Mixed flow turbines are somewhat of a cross between axial and radial flow turbines. Rajoo and Martinez-Botas [14] provided a comprehensive review of mixed flow turbine research and development up to the early 21st century. Thus, only important aspects as well as recent developments in mixed flow turbine research will be presented in this review.

In a mixed flow turbine, the inlet flow enters the rotor at an angle between the axial and radial direction. Hence the inlet flow has both axial and radial components. Figure 2.7

shows a schematic of a typical mixed flow turbine. The main difference between the mixed flow and the radial turbine is the cone angle, defined as the angle the blade leading edge makes against the radial axis in the z - r plane (as shown in Figure 2.7). For a radial turbine the cone angle is 90° by definition. The main advantage of a mixed flow turbine is possibility to have inlet blade angles other than zero, while still keeping the radial fibre requirement (also shown in 3D diagrams in Figure 2.7). This allows for mixed flow turbine to have peak efficiencies at a lower velocity ratio/higher specific speed (shown in Figure 2.8), while still maintaining the benefits of a radial turbine [14]. Thus a mixed flow turbine can perform better than a radial turbine under high load conditions, allowing it to more efficiently extract energy during the peak of the exhaust pulses from an ICE [14]. Other advantages of a mixed flow turbine mentioned by Rajoo & Martinez-Botas include lower secondary flows and separations due to lower curvature of flow path, a larger swallowing capacity, as well as lower wheel inertia.

Figure 2.7: Mixed flow turbine schematic and characterization [14]

The flow within a mixed flow turbine, unsurprisingly, shares characteristics with both axial and radial turbines. Kirtley et al. [15] showed through CFD simulations that the flow in a mixed flow turbine more closely resembles that in a radial turbine than an axial one. However, they also noted that their simulation results show a weak hub horseshoe vortex system that is typically a flow feature in axial turbines. Palfreyman and Martinez-Botas [16] also numerically compared the flow between a mixed flow and a radial turbine and also concluded that the flow in mixed flow turbines is similar to radial turbines. However, two main differences were highlighted. Firstly, there was a large region of high vorticity near the shroud of the radial turbine due to the relative

motion of the shroud wall as well as the 90° turn of the flow from radial to axial direction. This feature was not present in the mixed flow turbine. Secondly, there was a large separation vortex near the hub suction surface in the mixed flow turbine, which was not present in the radial turbine. While a hub horseshoe vortex structure was not mentioned by Palfreyman and Martinez-Botas, the hub suction side separation vortex might have originated from the suction side leg of the horseshoe vortex.

Figure 2.8: Typical axial, radial and mixed flow turbine efficiency against specific speed [6]

More recently, research on mixed flow turbines have shifted to focus more on their steady and unsteady performance for use in automotive turbocharger applications. Romagnoli and Martinez-Botas [17] developed a meanline model for predicting the steady state performance of both nozzleless and nozzled mixed flow turbines and found good accuracy when compared with experimental data. Chiong et al. then [18] developed a prediction model coupling 1-D gas dynamics with meanline model to predict the unsteady behavior of a single entry mixed flow turbine. They concluded that their prediction model was considered good, but was limited by the ability to experimentally measure all parameters instantaneously. Padzillah et al. [19] investigated the effects of turbine speed and pulse flow frequency on mixed flow turbine performance. They concluded that the optimum turbine speed resulted in the

highest instantaneous efficiency as well, while the relationship between pulse frequency and performance was less clear. The authors also pointed that simply averaging the flow parameters might lead to misleading results.

As computational speed increases, certain methodologies have become more viable in recent years. One such methodology is the usage of optimization algorithms for turbine design. Roclawski et al. [20] employed an evolutionary algorithm together with neural networks to optimize the design for a mixed flow turbocharger turbine wheel for maximum efficiency and minimum inertia at two design points. Khairuddin et al. [21] also employed genetic algorithm for optimizing a turbocharger mixed flow turbine wheel for maximum efficiency, and found that optimizing the hub and shroud profiles led to the largest efficiency improvement. Another area of interest is the effect of the stator on the performance of the rotor. Hakeem et al. [22] studied the effect of volute geometry on the steady and unsteady performance of a mixed flow turbine. Their results showed significant differences in the swallowing capacity, as well as differences in off-design performance of the rotor. By changing the design of the volute, they were able to obtain 3.24% increase in efficiency under steady flow and 5.8% increase in cycle-averaged efficiency during pulsating flow conditions. Rajoo and Martinez-Botas [23] compared the performance of straight and lean nozzle vanes for a mixed flow turbine, and found that at an optimized vane angle, lean vanes perform better than straight vanes at off-design points under steady conditions. However, straight vanes seem to perform marginally better than lean vanes under pulsating flow conditions.

2.3 Losses Mechanisms in Radial and Mixed Flow Turbines

The internal flow in turbomachinery is highly complex and three-dimensional. A great deal of research was undertaken to understand and minimize the internal flow structures and sources of aerodynamic losses in compressors and turbines. This led to improvements in aerodynamic efficiency. Over the decades, a greater understanding of the loss mechanisms within turbomachinery was achieved, but there are still significant gaps in knowledge even up till today.

Denton [24] provided a detailed summary of the state of research in turbomachinery loss mechanisms up to the early 1990s. Generally, turbomachinery losses can be broken

down into three main components: profile losses, tip leakage losses, and secondary/endwall losses. According to Denton, the three types of losses each contribute to roughly one third of the total turbomachinery losses. In addition, under off-design operating conditions, turbomachinery suffer from additional loss in efficiency due to non-optimum incidence angles. This is commonly known as incidence loss. For turbines, the exit kinetic energy of the turbine is normally unused and is thus considered a source of loss as well. This is normally called the exit loss, but will not be discussed in detail in this thesis.

Early methods of predicting turbomachinery losses rely on analytical and empirical models which were derived from experimental results. Numerous loss models have been developed by many authors in order to incorporate them into one-dimensional codes for turbomachinery performance predictions. The loss models usually deal with a specific type of loss (e.g. passage or tip leakage) independently, assuming no interaction between the various types of losses. As a result, the relative contribution of different loss sources could be represented easily, for example shown in Figure 2.9. Such a breakdown of loss sources allow a turbomachinery designer to focus more on losses that are more significant in order to improve the overall efficiency. However, this method might not accurately represent the flow in turbomachinery, as in reality there may be interactions between the types of losses, and this must be taken into consideration when proposing new designs or optimizations to turbomachinery. Moreover, empirical models were developed to fit existing experimental results, and care must be taken when using them outside the range of data tested in the original experiments. Indeed, it was mentioned by Denton [24] that “such correlations can tell us nothing about new design features that were not available at the time the correlation was developed”.

A review of the multitude of loss models available in literature is out of scope of this thesis, and only a few key papers that summarize the research on radial and mixed flow turbine performance prediction using loss coefficients are presented. Benson [25] conducted a review of numerous loss coefficient models up till 1970. He compared the loss predictions of the models to experimental data and concluded that turbine performance can be reasonably predicted using the suitable loss models and suitable exit angle values. More recently, Suhrmann et al. [26] reviewed several commonly used

loss models and compared their predictions to CFD results for a small sized radial turbine. They found that generally, available loss models provided satisfactory performance predictions for small sized radial turbines. It was noted, however, that all slip factor models overestimated the relative eddy in the rotor and the effect of rotor inlet area on slip factor was not taken into account. The authors also noted that Rodger's [27] secondary flow loss model overpredicted the losses. Persky and Sauret [28] conducted a similar exercise, but with much more loss models considered. Their study focused on on-design and off-design performance of radial turbines using non-ideal gases. More than 1.5 million different combinations of loss models were compared with CFD results, and the best combination achieved less than 3% difference from CFD results. These results showed that when appropriate loss correlations are chosen, one-dimensional models can reasonably predict turbine performance, and when combined with their fast computational speeds, can be a powerful tool for preliminary turbomachinery design.

Figure 2.9: Radial Turbine Loss Distributions: Left) Rohlik [29], Right) Huntsman & Hodson [30]

The improvements in computing power eventually made more complex numerical models viable. Two-dimensional (2D) methods began to be used in the 1960s [31], of which the most common type are streamline curvature methods. Many of these methods were based on Wu's [32] S1/S2 surface approach. A review of the numerical methods used up till the 1970s can be found in [33]. The fast advances in computing power through the late 1990s meant that 2D methods were quickly replaced, with three-dimensional (3D) inviscid methods being used in the 1970s, and 3D viscous models in

the 1980s [31]. Today, 2D streamline curvature methods are seldom used for analysis, though have been some efforts to use these methods as preliminary design tools instead (e.g. [34]).

3D CFD has become an indispensable tool for turbomachinery design and analysis. Today, advances in numerical methods, together with improvements in computational power, means that CFD is able to provide solutions for 3D, full-stage/multi-stage unsteady flows in reasonable amounts of time using high performance computing clusters. The advantages of CFD further extend to being able to study the internal flow structures of a turbine easily and in lots of detail. Compared to experiments, where sensors need to be installed at locations of interest (which are sometimes inaccessible), in CFD engineers can visualize and measure flow properties at any location, and also directly see streamlines, as seen in Figure 2.10. CFD can also be used to locate high entropy generation regions (e.g. Newton et al. [35]). The conveniences brought upon by modern CFD cannot be overstated. However, when using CFD for flow analysis, one should be mindful of the limitations of CFD. Denton [31] has highlighted several limitations of CFD, with turbulence and transition modelling being one of the most difficult to resolve. This was in fact already mentioned back in 1998 by Denton [36], showing that turbulence and transition modelling will for a long time be a limitation to the accuracy of CFD results.

Figure 2.10: CFD Post-processed flow visualizations

Unsteady simulation and higher fidelity methods have become more feasible due to increases in computational power. Several methods were highlighted by Tucker [37]. Of these, large eddy simulation (LES) and hybrid RANS-LES models have seen more

widespread use, often used in axial turbomachinery in areas where RANS turbulence modelling cannot accurately model the flow, such as jet engine low pressure turbines (LPT) where RANS models face difficulty in correctly predicting laminar to turbulent flow transition. Many examples of hybrid and LES use in turbomachinery can be found in [38] and [39]. As mentioned by Tucker, LES alleviates the problem of turbulence modelling by only modelling the smallest turbulent eddies and directly simulating the rest. This results in a more accurate simulation compared to unsteady RANS simulations, which allows for better understanding of the aerodynamics and the losses within the turbomachine. However, this comes at a cost of a massive increase in computational time required. It is mentioned by Gourdain et al. [40] that LES simulations are about 10000 times more expensive than RANS simulations.

Not much has been found in the literature on LES simulations in radial and mixed flow turbocharger turbines, the only publications coming from the KTH department of mechanics [41-43]. It was shown that when appropriate boundary conditions were provided (e.g. using measured data from experiments), LES simulations were much more accurate than unsteady RANS simulations for turbocharger turbines under pulsating flow conditions. This shows that LES can be used to better predict the unsteady flow structures in turbocharger turbines, which will help improve understand the loss mechanisms and provide insights on mitigating these losses and improving turbine efficiency under the unique operating conditions of an ICE.

Modelling uncertainties are still present in CFD, especially in RANS where all the turbulence is modelled. Even in higher fidelity models like LES, the small scale eddies (about 10%, [38]) are modelled. Therefore, experimental validation of CFD results are important to ensure that the CFD models used are appropriate and can sufficiently predict the real flow. Many experiments have been conducted over the past few decades measuring the performance of radial and mixed flow turbines, both steady and unsteady (e.g. [9, 17, 19, 29, 44-48]). Some experimental studies also included more detailed flow measurements which helped further understand the flow structures in radial and mixed flow turbines (e.g. [30, 49-53]). These studies provide valuable validation for numerical models to improve their accuracy.

In the following section, the mechanisms and research in each type of loss are presented separately. A quick browse at published literature will find that an overwhelming amount of research is dedicated to the fluid mechanics of axial flow turbomachinery. While in many cases the knowledge gained from axial turbomachinery research can be applied to radial machines as well, there are some very fundamental difference in flow structures in radial turbomachinery, and some of the methods used for axial turbomachinery in predicting or reducing losses might not be directly applicable to radial machines. To keep within the scope of this thesis, the review will focus mainly on the aspects that are more relevant and applicable to radial and mixed flow turbines.

2.3.1 Profile Loss

Profile loss arises mainly from the viscous dissipation in blade boundary layers away from the endwalls of turbomachinery. Away from the endwalls, the flow can be assumed to be two-dimensional, but for radial turbomachinery, this region might be small or non-existent due to the additional effects of rotation which skew the blade boundary layers. Denton [24] also includes the losses in blade trailing edges as profile losses, because these losses are also mainly two-dimensional in nature. Due to the relative simplicity of the loss mechanisms, the mechanisms of profile losses are well understood, and these losses can be predicted with considerable accuracy [24]. Most of the recent research on profile losses deals with correctly modelling transition and separation, and expanding the design space of turbines using these updated models.

For losses due to blade boundary layers, Denton mentioned that if the boundary layer is turbulent, the entropy generation due to the boundary layer is relatively independent of the boundary layer state, while the same cannot be said of laminar boundary layers [24]. This implies that predicting the location where transition occurs in the boundary layers is extremely important in accurately predicting boundary layer losses. Denton has also noted that in typical turbomachine operating conditions the Mach number has only a small influence on boundary layer losses. In Denton's formulation of blade boundary layer losses, the dominance of the suction surface in loss contribution is apparent. This should apply to radial turbomachinery as well, because the profile loss increases with blade surface velocity, and in radial turbomachinery the suction surface has higher blade surface velocity as well. The numerical results of Newton et al [35] on a mixed flow turbine seem to support this point, but hopefully more such analysis can

be done to obtain a clearer picture of the profile loss distribution of radial turbomachinery.

Efforts have been made to better understand and predict transition in axial turbomachinery, but transition in radial turbomachinery, in contrast, has not seen the same level of study as their axial counterpart. A survey through existing literature has failed to turn up any studies on transition onset and location in radial turbomachinery, but some insights could perhaps be gleamed from theory. The 90° passage turn in radial turbomachinery leads to significant differences in the flow fields between radial turbomachinery and their axial counterparts. The main source of these differences lie in the centrifugal and Coriolis forces present in radial turbomachinery. These forces affect the boundary layer characteristics and also contribute to secondary flow fields that are not typically found in axial turbomachinery. For axial turbomachinery, the Coriolis force introduces radially outward flows in the boundary layers, creating skewed three-dimensional boundary layers. For radial turbomachinery, the Coriolis force sets up both a pressure gradient and a velocity gradient in the tangential direction [54], and tends to have a major influence on blade boundary layer losses. The Coriolis force destabilizes the boundary layer on the pressure side, promoting boundary layer growth, and stabilizes the boundary layers on the suction side, significantly affecting the flow field in radial turbomachinery. The stabilizing effects of the Coriolis forces on the suction surface result in earlier onset of suction side flow separation in radial turbomachinery [55].

The Reynolds number has been shown to affect laminar-to-turbulent transition in two ways. The first is the transition location. Schreiber et al [56] conducted an experimental and analytical study on the effects of Reynolds number and freestream turbulence on transition in a compressor cascade and found that as Reynolds number increases, the location of transition moves upstream. The second effect of Reynolds number is in the separated region in separated-flow transition. It has been found by Volino [57] that at Reynolds number (based on suction surface length and exit velocity) higher than 150,000 the size of the separation bubble was very small and reattachment was almost immediate. This result was similar to that of Schreiber et al, in that as Reynolds number increases, the extent of the separation bubble decreases. On the other hand, at Reynolds numbers smaller than 50,000 there was no reattachment of the separation bubble. This

result agrees with that found of Praisner and Clark [58]. From these results, it is unclear whether increasing the Reynolds number by itself will cause profile losses to increase or decrease. While the earlier onset of transition to turbulent boundary layers increases the viscous dissipation, the smaller extent of the laminar separation bubble will act to reduce losses.

The amount of freestream turbulence also affects the blade boundary layer losses. Experimental results from Schreiber et al [56] show that at high Reynolds numbers ($Re = 2.0e^6$), increasing the turbulence levels causes transition to move upstream, and at the highest turbulence levels tested, bypass transition was induced. Douglas et al [59] showed that through increasing the freestream turbulence level resulted in lower losses in a linear transonic compressor cascade, and attributed the loss reduction to reduced laminar separation on the suction surface. These results seem to suggest that laminar separation bubbles contribute to a significant amount of blade boundary layer losses, and eliminating these separation bubbles is key to reducing profile losses, and more quantitative studies are needed to assess the effects of freestream turbulence on blade boundary layer losses.

Another important factor influencing blade boundary layer losses is surface roughness. Bons [60] conducted a comprehensive review on surface roughness effects in gas turbines, and the main conclusion to be drawn is that for both compressors and turbines, at low Reynolds numbers, surface roughness can reduce blade boundary layer losses through eliminating laminar separation bubbles. However, at high Reynolds numbers, where the boundary layers are already turbulent, surface roughness will lead to increased boundary layer thickness and induce turbulent separation, resulting in increased losses. A more recent study by Im et al [61] also showed similar results. Bellucci et al [62] conducted a numerical study on roughness effects on a steam turbine, and found an increase in losses as surface roughness increases, similar to previous studies at higher Reynolds numbers. However, their minimum Reynolds number tested was probably too high to show the performance benefits at lower Reynolds numbers.

The blade trailing edge loss is the other major component of profile loss, and the main cause of this loss is the mixing out of the trailing edge wake. The trailing edge losses contribute about a third of the total profile losses [24], and in transonic conditions

trailing edge losses in turbines can contribute up to 70% of overall loss [63]. Thus, trailing edge losses cannot be neglected. However, despite understanding how the losses occur, prediction of the magnitude of trailing edge losses without using empirical data is still a challenge. One difficulty noted by Denton in predicting trailing edge losses is the inability to predict the base pressure for the trailing edge losses [24]. The trailing edge loss increases rapidly as Mach number increases.

There is relatively sparse literature covering trailing edge losses in turbomachinery. The Von Karman Institute has published a series of papers investigating the flow on the trailing edge of an axial turbine blade [64-66] and their results show that Mach number is a very important factor in the trailing edge base pressure distribution. As the Mach number increases from low subsonic to high subsonic region, the isobaric base pressure region gets replaced by a non-uniform pressure distribution with a minimum at the centre of the trailing edge, which then becomes an isobaric pressure region again at low supersonic Mach numbers. This means that the trailing edge losses increase as Mach number increases. Also, the base pressure decreases as Mach number increases [66]. However, the applicability of this for blades other than those with low suction side rear curvature is unknown, and more investigations of this type are needed, in hopes of a general correlation for base pressure. Xu and Denton [63] also found experimentally that trailing edge losses increase in proportion to the thickness of the trailing edge. Overall, more studies are needed for better modelling of trailing edge losses.

The overall profile losses are highly influenced by the state of the boundary layer on the blade, and this will be determined by several factors such as certain blade geometrical parameters and performance parameters. Numerous studies have attempted to include these parameters in profile loss coefficients, both as a way to predict the losses during preliminary design, and also possibly as a design tool to minimize profile losses. Coull and Hodson [67] developed an empirical method of predicting profile losses in high-lift low pressure turbines which focuses on predicting the suction surface boundary layers accurately. In their paper, the profile losses are determined mainly by the boundary layer integral parameters on the trailing edge suction surface, which can be estimated from design parameters such as the diffusion ratio, non-dimensionalized peak velocity location on suction surface, leading edge loading, Reynolds number, reduced frequency, and the inlet and exit flow angles. Suhrmann et al [26] numerically

validated several loss models, which include models for skin friction losses, for small radial turbines. They found good agreement in skin friction losses between CFD and the loss models tested.

From the above, it can be seen that profile losses are determined mainly by the boundary layer characteristics, and the accuracy of profile loss models are largely dependent on accurately modelling boundary layer parameters, especially in conditions like laminar-to-turbulent transitions and separation bubbles, where there is still work to be done in terms of modelling. More work is also needed to better quantify and model trailing edge losses.

2.3.2 Tip Leakage Loss

Tip clearance loss, also known as tip leakage loss, is another major component of loss in a turbomachine. Denton estimated tip leakage losses to contribute about a third of total losses, but for mixed flow turbines leakage loss might contribute more than half of the total entropy generation [35]. Tip leakage loss is caused by flow leaking over the blade tip gap, driven mainly by the pressure difference between the pressure and suction surfaces [68]. Denton [24] has clearly outlined the main loss mechanisms in tip leakage flow. For unshrouded blades such as turbocharger turbine blades, the thickness of the blade relative to the tip gap also affects the loss mechanism, as shown in Figure 2.11.

Tip leakage flow in turbines results in a decrease in the mass flow through the blade passage, and the blade lift is reduced as well. The leakage flow enters the tip gap from the pressure surface, and exits from the suction surface, causing losses [24]. The first source of tip leakage loss is the loss over the tip gap itself. For unshrouded turbine blades, the leakage flow separates from the blade tip and contracts into a jet over the separation bubble, which may or may not reattach depending on the blade thickness. This loss is usually small compared to the mixing losses between the leakage flow and the main flow.

As the leakage flow exits from the suction surface, further mixing occurs between the leakage flow and the main flow, resulting in further entropy generation, mainly near the suction surface of the blade [24]. As the leakage flow exits from the suction surface and meets the main passage flow, the difference in velocities between the two flows results

in the formation of a leakage vortex. However, it is to be noted that the leakage vortex does not necessarily generate losses, and the main loss mechanisms are due to the mixing processes that occur in the leakage flow [24].

Figure 2.11: Tip leakage loss on unshrouded blades [24]

More recently, there has been additional attention paid to the loss mechanisms through leakage vortex breakdown. While the leakage vortex itself is an inviscid phenomenon, the breakdown of the leakage vortex has been shown to generate additional losses. In a review by Yang et al [68] (in Chinese), it was mentioned that the details on how the leakage vortex grows and travels are still unclear, as well as the mechanisms of leakage vortex breakdown, especially in radial turbomachinery. Yamamoto [69] investigated the loss mechanisms close to the endwalls of a linear turbine cascade and found that interactions between the leakage flow and the passage vortex creates high losses on the suction surface. More recently, it has been found by Huang et al. [70] that a positive pressure gradient decelerates the tip leakage vortex core, and this can either increase or decrease the mixing losses of the vortex, depending on the swirl number and vortex core velocity ratio. If the increase in pressure causes the vortex flow to stagnate, then vortex breakdown occurs, resulting in high losses. Further studies of axial turbine tip leakage vortex breakdown showed that the vortex breakdown is affected by the initial

streamwise velocity of the vortex core, as well as the pressure gradient (which is influenced by blade loading). These parameters can be influenced by the inlet flow velocity profile [71], as well as the amount of blade rotation [72].

The relative motion between the shroud and the blade has been shown to decrease the amount of leakage flow [73-76]. This is because the shroud moves from the suction side to the pressure side relative to the blade, producing a scraping effect which drags fluid from the suction surface to pressure surface against the direction of the leakage flow. This is illustrated in Figure 2.12. However, it is important to note here that even though tip leakage flow is reduced, Tallman and Lakshminarayana [74] had found that overall losses did not decrease, as there was a corresponding increase in losses due to near-casing secondary flow. Another factor affecting the overall losses might also be that relative casing motion increases the losses due to leakage vortex breakdown, as shown in a study by Gao et al [72]. The scraping effect due to the relative casing motion is more significant in radial turbomachinery, due to the larger radius of the casing [77]. Dambach et al. [53] attributes this as the main reason why tip leakage losses are generally lower in radial turbines than axial ones for the same gap height. A more detailed numerical study on a radial turbine by Deng et al. [78] further revealed that the magnitude of the scraping effect and the pressure difference between the pressure and suction side are the dominant factors affecting the leakage flow. At very tight tip clearances ($< 0.5\%$ channel height), Maesschalck et al. [79] found out that the scraping effect actually causes reverse leakage flow in the tip gap region of a high speed axial turbine.

Figure 2.12: Flow Over Blade Tip at: Left) 10% Meridional Chord, Middle) 50%, Right) 90% [78]

For the losses occurring over the tip gap, it has been found that the contraction coefficient of the leakage jet is influenced by the radius of the pressure surface corner, and this will result in differences in the loss magnitudes within the tip gap [24]. Heyes

et al [80] suggest that the radius of the pressure surface corner should be kept small, because although rounding the pressure surface corner will reduce losses within the tip gap, it leads to higher overall losses through an increase in the discharge coefficient. The thickness of the blade itself will also determine if the leakage jet reattaches to the blade. If the blade is thick relative to the tip gap, as is usually the case for turbines, the leakage jet then reattaches and mixes out within the gap, generating entropy. If the blade is thin, there is no reattachment of the leakage jet within the tip gap. None of the factors will significantly affect the overall tip leakage loss magnitude, since the loss within the tip gap is only a small fraction of the overall tip leakage loss.

Much more attention has been paid to the mixing loss between the leakage and main flow. From the control volume model of a leakage jet mixing with the main flow at an angle, the angle between the leakage jet and the main flow can significantly influence the mixing losses. Since the direction of the main flow will be in the streamwise direction, the streamwise component of the leakage jet velocity will affect the amount of mixing losses. To minimize the mixing losses, the difference in streamwise velocity between the leakage flow and the main flow should be as small as possible [24], and the angle between leakage flow and main flow should be kept to a minimum as well [81]. Blade loading is another factor that influences the tip leakage flow, and in turn influences the leakage losses [24]. This is not surprising, since the tip leakage flow is driven by the pressure difference between the pressure and suction surface of the blade, and changes to this pressure difference and distribution will affect the leakage flow and its associated losses.

Storer and Cumpsty [81] studied the leakage flow mechanisms in an axial compressor and established some important parameters affecting the leakage loss production. One of the most important is the ratio between the tip gap and the blade height, which was found to be almost directly proportional to the leakage loss at practical tip gap ranges. They also found out that the degree of reaction has some effect on the leakage loss, while turbulence modelling and relative endwall motion should have little effect on the magnitude of leakage loss. However, at low tip gap ranges, there appear to be an optimum tip gap which gives the minimum loss. This is supported by a numerical study from Sakulkaew et al. [82], which found that there is an optimum efficiency point at a tip clearance of less than 0.8%, which varies with different operating points. Sakulkaew

et al.'s study also showed an almost linear correlation between tip gap and leakage loss at tip gaps between 0.8% to 3.4%, which agrees with Storer and Cumpsty's results. At larger tip gaps, Sakulkaew et al.'s results showed that the leakage loss becomes less sensitive to changes to tip gap. The previous studies were conducted on compressors, but the findings should reasonably apply to turbines as well.

There have been some numerical investigations on the parameters affecting losses through the tip leakage vortex. Huang et al [70] found that for a positive pressure gradient along the streamwise direction, if the vortex has a large swirl number and core velocity ratio, typically found in turbines, the vortex will show more vortex-like behavior and mixing losses will decrease as a pressure rise increases. The above analysis is for pressure rises that are not large enough to cause vortex breakdown.

For leakage vortex breakdown losses, the main factors affecting the amount of loss is the location and the intensity of the breakdown [72]. One of the factors affecting vortex breakdown location is the suction surface pressure gradient, as a larger adverse pressure gradient will result in the leakage flow reaching a stagnation point earlier, thus resulting in a more upstream breakdown of the leakage vortex. Therefore, blade loading distribution is an important design parameter that affects the amount of mixing losses caused by vortex breakdown, and reducing the adverse pressure gradients near the trailing edge can potentially result in lower vortex breakdown losses [70]. Another factor that affects vortex breakdown location is the initial streamwise velocity of the vortex core. It has been found by Gao et al [71] that a low initial streamwise velocity of the vortex core will result in a more upstream vortex breakdown location, and the converse will result in vortex breakdown occurring further downstream. The initial streamwise velocity of the vortex core has been found to be increased by negative inlet flow incidence angles [71], as well as lower relative casing motion speed [72], resulting in a more downstream leakage vortex breakdown location. Conversely, increasing the flow incidence angle [71] and a higher relative casing motion speed [72] reduces the initial vortex streamwise velocity and result in a more upstream leakage vortex breakdown location, which increases the vortex breakdown losses.

The vortex breakdown intensity has been shown by Gao et al. [72] to be related to the minimum streamwise velocity of the vortex core. Their numerical results show that the

vortex breakdown intensity increases as the value of the minimum streamwise velocity near the vortex core becomes more negative. This minimum streamwise velocity is in turn affected by the rotational speed of the blade, through the relative casing motion, as well as Coriolis and centrifugal forces. The increase in vortex breakdown intensity will lead to higher vortex breakdown losses. The results show some important details of tip leakage vortex breakdown, but more investigations, both numerical and experimental, are needed to gain a more complete picture of the tip leakage vortex breakdown mechanisms and the associated losses. Experimental data will be extremely valuable in validating the numerical results.

2.3.3 Secondary/Endwall Loss

Secondary or endwall loss, is another significant source of loss in a turbomachine. Secondary losses arise near the annulus walls of the turbomachine. This loss arises from interaction between secondary flow structures and the boundary layers near the endwall, and also a loss in secondary kinetic energy [83] , and is less well understood and predictable as compared to tip leakage and profile losses.

There have been plenty of research on secondary losses on axial turbines. Early cascade data and correlations on endwall flow structures were summarized in a review conducted by Dunham [84], and it was found out that many of the correlations do not agree well with each other, or even with experimental cascade data. Over the next decade or so, more research was conducted to study the endwall flow structures in detail. The results of the research were summarized by Sieverding [85]. From the research conducted, some important endwall loss mechanisms were summarized. These include: Increase in inlet endwall boundary layers until separation, separation bubble near the blade leading edge, growth of new boundary layer after separation, corner losses, shear stress effects along separation lines, shearing between the passage vortex and suction surface of the blade, mixing between crossflow and blade flow, and vortex dissipation and mixing of flow downstream of cascade. The results of these studies reveal important insights on understanding and predicting endwall losses, but the highly three-dimensional nature of the flow still makes quantitative prediction of endwall loss a large problem.

By the 1990s the secondary flow structures in axial turbine cascades have been reasonably well understood. The basic flow structure of secondary flow in an axial turbine cascade, as proposed by Sharma and Butler [86] is illustrated in Figure 2.13. The structure, formation and mechanisms of these secondary flow structures are explained in detail by Denton [24]. However, there was still some disagreement on the development of the suction side leg of the horseshoe vortex, as mentioned by Cui et al. [87]. LES simulation by Cui et al. showed that the suction side leg travelled above the passage vortex, eventually merging with the passage vortex at the trailing edge, shown as vor_A on the right side of Figure 2.13. This is different from the structure proposed by Sharma and Butler.

Figure 2.13: Secondary Flow Structures in Axial Turbines: Left) Schematic [86], Right) LES Simulation [87]

Even though the basic secondary flow structures and mechanisms are relatively well understood, the endwall losses arising from the interactions between the secondary flow structures and the boundary layers at the endwalls cannot be accurately predicted. Most existing formulation to predict endwall loss rely on empirical correlations, such as those by Dunham [84] and Sharma and Butler [86]. Following Sieverding's review, there have been research conducted to study in more detail the flow structures that affect endwall losses. Langston [88] summarized the progress in endwall flow research up to the beginning of the 21st Century, and important breakthroughs made during this period was the prediction of the inlet boundary layer separation using a simple correlation, which marks a step closer to predicting the endwall flow properties. Langston also suggested the use of entropy generation, previously mentioned by Denton [24], as a measure of loss. This, coupled with the increasing use of 3D Navier-Stokes solvers, can offer new insights into the endwall loss mechanisms. With the increase in computing power, this approach has become much more practical, and was attempted by Denton

and Pullan [83]. They split the computational domain into different regions, and calculated the entropy generated in each region. Thus, the endwall loss can be determined from the total entropy generation near the endwalls. The contribution of endwall loss to the total loss can also be determined through this method.

While advancements were made in understanding secondary losses in axial turbines, there are some differences in the secondary flow in radial and mixed flow turbines. Denton [24] has mentioned some basic differences in the flow in radial turbomachinery, due to the turn in the flow passage. This results in stronger and different secondary flow structures in radial and mixed flow turbines. At the inlet region of a radial turbine, where the flow is predominantly in the radially inward direction, the Coriolis force acts in the direction of flow rotation. This is in the opposite direction of the pressure gradient between the pressure and suction surfaces. Near the shroud, the scraping flow is also in the opposite direction as the turbine rotation. Thus, the overall blade to blade flow structure should be dependent on the relative magnitude of these 3 factors. Baines [89] attributes the strong cross-passage Coriolis force to be the main reason for the pressure-side separation typically found in radial turbines at large negative incidence angles. Numerical simulations by Palfreyman and Martinez-Botas [16] show that for a radial turbine, the Coriolis force is strong enough that the cross-passage flow is already predominantly in the direction of rotation by 10% chord even when the inlet flow has a negative incidence angle (i.e. opposing direction of rotation). On the other hand, for the mixed-flow turbine investigated by Palfreyman and Martinez-Botas, only the flow near the hub is predominantly influenced by the Coriolis force, creating a large counter-clockwise (when viewed downstream towards trailing edge) passage vortex structure. This flow feature is also captured in a numerical simulation of a mixed-flow turbine by Chen et al. [90]. For a radial turbine, the flow near the shroud apparently then rolls up into a vortex covering the full pitch of the blade passage, due to a combination of the scraping flow and the 90° turn in the meridional flow path [16], which leads to high entropy generation. This was not found in the mixed flow turbine simulations [16, 90]. The flow features described above are reproduced in Figure 2.14 for reference. The results from Chen et al. also showed the formation of a horseshoe vortex, of which the pressure side leg then migrates to the suction surface, causing suction side flow separation near the hub. Their results also show complex flow structures on the suction surface, shown in Figure 2.15.

Figure 2.14: Flow Near Leading Edge. Left and Middle: Passage Secondary Flow Direction and Vorticity Contours (Palfreyman & Martinez-Botas [16]), Right: Passage Limiting Streamlines (Chen et al. [90])

Figure 2.15: Limiting Streamlines on Leading Edge Hub and Blade Suction Surface, Chen et al.[90]

Nearer to the exducer section of the turbine rotor, the meridional flow path will be in the axial and circumferential direction, and thus the secondary flow paths are expected to more closely resemble that of axial turbines. The Coriolis force is now in the outward radial direction, and thus secondary flow from the hub towards the shroud is expected. This, coupled with the cross passage flow from pressure to suction side driven by the pressure gradient, gives rise to complex secondary flow in this region [89]. However, the simulation results from both Palfreyman and Martinez-Botas, and Chen et al. show that hub to shroud secondary flow occurs near the mid-chord section, and near the trailing edge the dominant secondary flow is cross passage flow from the pressure to suction side. Chen et al. attributes the mid-chord secondary flow to be due to the centrifugal force due to the curvature of the passage. The results from Chen et al. also

shows that near the hub there is still secondary flow caused by the Coriolis force even near trailing edge, which is not present in Palfreyman and Martinez-Botas's results. These inconsistent results suggest that it is difficult to generalize the secondary flow structures in radial and mixed flow turbines, but with more data from different turbines some correlations regarding the relative magnitudes of the contributing factors to the secondary flows might be derivable. Unfortunately, such data is currently quite sparse in literature, and almost all the available data are from numerical simulations, in which the validity of the models (especially turbulence and transition) will always be questionable. Analytically, one can expect that the Coriolis force and scraping flow will be influenced by the rotor radius and the rotation speed, the passage cross flow by the blade loading. The centrifugal force will be influenced by the rotor radius, radius of curvature of the meridional flow path, and the rotational speed as well.

2.4 Methods to Reduce Turbomachinery Losses

2.4.1 Profile Loss

There has been some work found in literature related to reducing profile losses. Wisler [91] designed an airfoil with a custom tailored surface boundary layer which aims to prolong the onset of laminar-to-turbulent transition, as well as minimize the momentum and displacement thickness at the suction surface trailing edge. Wisler found this method effective in reducing profile losses, but has the drawback of increasing flow separation at the hub at flow rates below design conditions. More recently, Bloxham and Bons [92] attempted to simultaneously reduce both profile and endwall losses in an axial turbine by utilizing both leading-edge suction and vortex generator jets. Their results on a linear turbine cascade show that the vortex generator jets decreased the wake total pressure losses by 68% at low Reynolds numbers, where profile losses dominate the wake losses. It is also found from their study that leading edge suction provides minimal additional benefits in loss reduction.

2.4.2 Tip Leakage Loss

The most straightforward way of reducing tip leakage loss is by decreasing the gap between the blade tip and the shroud, but obviously there is a limit to the minimum gap that can be practically achieved. This constraint on tip clearance greatly impacts small turbocharger turbines. For small turbomachinery such as those found in automotive

turbochargers, the tip gap practically achievable might be around 3% blade height. Therefore, other methods in reducing the tip leakage loss through flow control are needed. Existing literature on reducing tip leakage losses is substantial, but most of them deal with axial turbomachinery. Corresponding literature for radial turbocharger turbines are much scarcer. Deng et al. [93] proposed using circumferential slots, axial slots or honeycombs on the shroud surface to reduce the tip leakage flow for radial inflow turbines, but no detailed investigation was done to access the effectiveness of these measures. For radial turbines, since the scraping effect helps to block tip leakage flow nearer to the leading edge, designing blades that are more front loaded might help in mitigating tip leakage losses.

Geometry modification is a popular passive method of reducing tip leakage loss as they are relatively simple to implement. Geometry modifications can be applied to the rotor blade, or to the endwalls of a turbomachinery. Several different methods of geometrical modifications to the turbomachinery blade have been investigated in literature, some of which shown in Figure 2.16. One of the earlier methods was tailoring rotor airfoils to change the blade loading distribution so as to reduce losses. This was done by Wisler [91], through low-speed compressor model testing, in which he found that front-loaded airfoils were able to reduce tip leakage losses. Squealers, which are depressions on the tip of the airfoil, have also been widely investigated in literature. Heyes et al. [80] investigated the effects of both pressure side and suction side squealers on axial turbine cascades, and found suction side squealers to be especially effective in reducing the cascade losses. Later studies, such as those of Schabowski and Hodson [94] also showed reduction of losses through the usage of squealers. In particular, Schabowski and Hodson's results show that a cavity type blade tip (both pressure and suction surface squealers) is more effective than just a suction side squealer, and the benefits of squealers increase as the tip gap increases. From Schabowski and Hodson's numerical studies, the squealer prevents the reattachment of leakage flow over the blade tip, similar to how leakage flow behaves over a thin blade. This results in a lower discharge coefficient, and thus lower tip leakage losses. Lee and Chae [95] experimentally investigated the effect of squealer rim height on tip leakage losses, and found that increasing squealer rim height weakens the tip leakage vortex and reduces the total pressure loss in a high-turning turbine. However, this is only effective up to a squealer rim height of 5.5% chord length. Similar results were obtained by Zhou and

Hodson [76], who also found that reducing squealer thickness reduces tip leakage loss. Zhou and Hodson noted however that there is a complex relationship between the combination of these two effects, and that for higher squealers decreasing the squealers is less beneficial aerodynamically.

The winglet is another kind of geometrical modification to the rotor blade. Harvey and Ramsdan [96], based on a review of previous research, derived a winglet design which includes a cavity (squealer). The aim of their study is to reduce tip leakage losses while being smaller than a full shroud. The numerical simulations suggest that an efficiency improvement of 1.2-1.8% can be achievable at a tip gap of 2% blade span. Lee et al [97], however, found only marginal improvements in turbine efficiency using pressure-side winglets of different width. The winglets weakened the tip leakage vortex, and the tip leakage losses, but intensified the passage vortex and its associated losses. Schabowski and Hodson [94] tested several winglet designs, including winglets with squealers similar to those of Harvey and Ramsdan (see Figure 2.16). Their results showed that the plain winglets (without any squealers) provided only marginal benefits over a flat blade tip, while the most substantial improvements came from winglet designs with squealers. The results suggest that squealers are much more effective in reducing overall losses than winglets, which may explain the apparent contradictory results between Harvey and Ramsdan and Lee et al.

Figure 2.16: Geometry Modifications: Left) Winglet and Squealer [94], Right) Shroud Endwall Contouring [98]

Endwall modifications were also investigated by several researchers. The first type of endwall modification is casing modification, which is commonly investigated in axial compressors, but have found some use in turbines as well. Wei et al. [98] numerically investigated a contoured axisymmetric casing design on an axial turbine (see Figure

2.16). The design features an arc-like cavity on the casing, with the blade tip shaped to follow the arc. They found out that the design reduces the amount of tip leakage flow, and also reduces the entropy generation near the blade tip. For shrouded turbines, Rosic and Denton [99] explored using vanes in the shroud exit cavity to align the leakage flow to the mainstream flow direction. By varying the geometry and spacing between the vanes, they were able to obtain a 0.4% increase in efficiency. Deng et al. [78] explored using cavity-like seals on a radial turbine. Their simulations showed that the best configuration increased power output by 1.54% and efficiency by 0.26%. The results showed that there is a trade-off between decreasing the pressure difference between pressure and suction side (which reduces power output) and reducing tip leakage flow (and thus leakage losses), and that certain configurations might provide a net performance improvement. More studies in this direction might help establish some correlations on this trade off.

In addition to geometry modifications, there were also attempts to mitigate tip leakage loss by controlling the tip leakage vortex directly. Gao et al. [100] believed that methods that reduce tip leakage losses through reducing the total tip leakage flow have already reached their limit, and new methods should focus on controlling the leakage flows to reduce losses. Therefore, they proposed using tip injection on the blade to control the tip leakage vortex, and achieved some improvements in turbine efficiency. Behr et al. [101] injected cooling air through the casing in a high-work axial turbine, shown in Figure 2.17. The flow was injected with a radial and circumferential flow component, and acts like a ‘curtain’ which cuts into the main passage flow. It was found that the injection was able to reduce both the size and intensity of the rotor tip leakage and tip passage vortices, and improve the turbine isentropic efficiency by 0.55 percentage points. Wang et al. [102] investigated using cooling flow injection over a honeycomb blade tip (see Figure 2.17), and found reduced pressure losses using the method. However, no efficiency values were given.

Studies were also conducted to explore the usage of wake flow from blade rows upstream to reduce the tip leakage losses. Qi and Zhou [103] showed that by using the upstream periodic wake flow to control the leakage flow, the tip leakage vortex strength was reduced. The results show a reduction in the entropy gain in the turbine, and an increase in turbine efficiency.

Figure 2.17: Left) Casing Injection, Behr et al. [101] Right) Honeycomb Tip Injection, Wang et al. [102]

2.4.3 Secondary/Endwall Loss

Several methods exist in literature that attempt to reduce the secondary losses in both axial turbines and axial compressors, some of which are shown in Figure 2.18. Three-dimensional blade designs were one method mentioned in literature [104] to be able to reduce endwall losses. Weingold et al. [105] used bowed stators (compound blade lean) as a form of 3D blade design on axial compressors and found that efficiency improvements of more than one percent can be achieved over a wide operating range. Duden et al. [106] also investigated using blade lean near the shroud surface, combined with endwall contouring, to improve turbine performance. They achieved a reduction in the endwall losses, but this was offset by higher inlet boundary layer losses and profile losses. Despite the results obtained, Denton & Xu [104] mentioned that the exact mechanisms on how blade lean reduces endwall loss is not clear.

Modifying airfoil shapes have also been a method investigated in literature. Wisler's [91] low speed model testing on compressors found that airfoils that have loaded leading edges and unloaded trailing edges have reduced hub flow separation, which can lead to reduced hub endwall losses. The results were contradictory to that of axial turbines, where front loaded blades have higher endwall losses [107, 108]. Sauer et al. [109] introduced a 'bulb' on the leading edge (shown in Figure 2.18) which they found reduced the secondary losses by about half. They found that the bulb intensified the suction side leg of the horseshoe vortex, which moves the counter-rotating passage vortex away from the profile boundary layer on the suction surface and deforms it.

Endwall contouring is another popular method used in reducing endwall losses. Duden et al [106] mentioned that lowering the endwall pressure using contouring can help decrease the radial pressure gradient towards mid span on the suction side. This helps

to obstruct the radial movement of secondary flow and reduce secondary losses. Other methods of endwall contouring generally also include some form of modifying the pressure field at the endwall to improve the flow field, reducing losses. Praisner et al [108] and Turgut & Camci [110] (shown in Figure 2.18) have both shown reductions in endwall losses using endwall contouring. Sharpar et al. [111] included real geometry effects in an optimization of endwall contouring for a high-pressure turbine stage, and achieved reduced secondary loss and an overall efficiency improvement of 0.43%.

Figure 2.18: Reducing Endwall Losses: Left) LE Bulb (Sauer et al., 2001 [109]), Right) Enwall Contouring (Turgut & Camci, 2015[110])

Besides the methods mentioned above, active flow control techniques are also used to reduce endwall losses. One such example is injection of vortex generator jets [112] to reduce the endwall losses. Bloxham & Bons [92] also included a leading edge suction in combination with the injection flow, but found that although the suction removed the horseshoe vortex structure completely, the total wake losses did not decrease significantly. From the results, they concluded that the horseshoe vortex system does not contribute much to the generation of endwall loss. Other methods of reducing endwall losses include endwall fencing. Kawai et al [113] used endwall fences on an axial turbine cascade, and found that fences were able to trap the pressure side leg of the horseshoe vortex, and reduce the endwall losses.

Despite the diversity of methods in literature, however, it was mentioned by Denton & Pullan [83] that there is no universally accepted method that provides consistent reductions in endwall loss in turbines. There are also studies on taking advantage of unsteady flow interactions in the turbine cascade to reduce the endwall losses [114].

More knowledge on the mechanisms behind reduction in endwall losses need to be obtained and understood in order for better design for turbines with higher efficiency.

For radial and mixed flow turbines, the author has not yet come across any literature with regards to secondary loss reduction. Some of the methods used in axial turbines, such as endwall contouring and active flow control methods might be effective, but detailed investigations are needed.

References

- [1] A. Buechi, "Improvements in Combined Internal Combustion Engines and Turbines," United Kingdom Patent GB190624980A, 1905.
- [2] ABB Turbo Systems Ltd. (2005) A Century of Turbocharging. *Turbo Magazine* Available: http://www.swiss-ships.ch/berichte-buecher/berichte/ABB_Turbo_Systems/TM_2005_100Y_EN.pdf
- [3] Garrett-Advancing Motion, "Garrett Motion Inc. Investor Conference Executive Overview," ed, 2018.
- [4] N. Watson and M. S. Janota, *Turbocharging the Internal Combustion Engine*. Palgrave Macmillan, 1982.
- [5] Turbo Dynamics. (2019). *How Does a Turbocharger Work? | Turbodynamics.co.uk*. Available: <https://www.turbodynamics.co.uk/technical/understanding-turbochargers/>
- [6] A. Whitfield and N. C. Baines, *Design of Radial Turbomachines*. Longman Scientific & Technical, 1990.
- [7] A. J. Feneley, A. Pesiridis, and A. M. Andwari, "Variable geometry turbocharger technologies for exhaust energy recovery and boosting-A Review," *Renewable and sustainable energy reviews*, vol. 71, pp. 959-975, 2017.
- [8] A. Pesiridis and R. F. Martinez-Botas, "Experimental evaluation of active flow control mixed-flow turbine for automotive turbocharger application " *Journal of Turbomachinery*, vol. 129, no. 1, pp. 44-52, 2007.
- [9] S. Rajoo, "Steady and pulsating performance of a variable geometry mixed flow turbocharger turbine," Doctor of Philosophy, Imperial College London, 2007.
- [10] K. Cao, "The Development of a Pulse-optimized Flow Control Method for Turbocharger Turbine Performance Improvement," Doctor of Philosophy, Imperial College London, 2017.
- [11] E. A. Baskharone, *Principles of turbomachinery in air-breathing engines*. Cambridge University Press, 2006.
- [12] C. Hall and S. L. Dixon, *Fluid mechanics and thermodynamics of turbomachinery*. Butterworth-Heinemann, 2013.
- [13] S. Quoilin, M. V. D. Broek, S. Declaye, P. Dewallef, and V. Lemort, "Techno-economic survey of Organic Rankine Cycle (ORC) systems," *Renewable and Sustainable Energy Reviews*, vol. 22, pp. 168-186, 2013.
- [14] S. Rajoo and R. Martinez-Botas, "Mixed flow turbine research: A review," *Journal of Turbomachinery*, vol. 130, no. 4, p. 044001, 2008.
- [15] K. R. Kirtley, T. A. Beach, and C. Rogo, "Aeroloads and secondary flows in a transonic mixed-flow turbine stage," *Journal of turbomachinery*, vol. 115, no. 3, pp. 590-600, 1993.
- [16] D. Palfreyman and R. F. Martinez-Botas., "Numerical study of the internal flow field characteristics in mixed flow turbines," in *Proceedings of the ASME Turbo Expo*, Amsterdam, 2002.
- [17] A. Romagnoli and R. Martinez-Botas, "Performance prediction of a nozzled and nozzleless mixed-flow turbine in steady conditions," *International Journal of Mechanical Sciences*, vol. 53, no. 8, pp. 557-574, 2011.
- [18] M. S. Chiong, S. Rajoo, A. Romagnoli, and R. F. Martinez-Botas, "Single entry mixed flow turbine performance prediction with 1-D gas dynamic code coupled with mean line model," *International Journal of Gas Turbine, Propulsion and Power Systems*, vol. 4, no. 2, pp. 8-16, 2012.

- [19] M. H. Padzillah, S. Rajoo, and R. F. Martinez-Botas, "Influence of speed and frequency towards the automotive turbocharger turbine performance under pulsating flow conditions," *Energy Conversion and Management*, vol. 80, no. 416-428, 2014.
- [20] H. Roclawski, M. Böhle, and M. Gugau, "Multidisciplinary design optimization of a mixed flow turbine wheel," presented at the ASME Turbo Expo 2012: Turbine Technical Conference and Exposition, 2012.
- [21] U. Khairuddin, A. W. Costall, and R. F. Martinez-Botas, "Influence of geometrical parameters on aerodynamic optimization of a mixed-flow turbocharger turbine," presented at the ASME Turbo Expo 2015: Turbine Technical Conference and Exposition, 2015.
- [22] I. Hakeem, C. C. Su, A. Costall, and R. F. Martinez-Botas, "Effect of volute geometry on the steady and unsteady performance of mixed-flow turbines," *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, vol. 221, no. 4, pp. 535-549, 2007.
- [23] S. Rajoo and R. F. Martinez-Botas, "Lean and straight nozzle vanes in a variable geometry turbine: a steady and pulsating flow investigation," presented at the ASME Turbo Expo 2008: Power for Land, Sea, and Air, 2008.
- [24] J. D. Denton, "The 1993 igt scholar lecture: Loss mechanisms in turbomachines," *Journal of Turbomachinery*, vol. 115, pp. 621-656, 1993.
- [25] R. S. Benson, "A review of methods for assessing loss coefficients in radial gas turbines," *International Journal of Mechanical Sciences* vol. 12, no. 10, pp. 905-932, 1970.
- [26] J. F. Suhrmann, D. Peitsch, M. Gugau, T. Heuer, and U. Tomm, "Validation and development of loss models for small size radial turbines," in *ASME Turbo Expo 2010: Power for Land, Sea, and Air*, 2010, pp. 1937-1949: American Society of Mechanical Engineers.
- [27] C. A. Rodgers, "A cycle analysis technique for small gas turbines," in *Proceedings of the Institution of Mechanical Engineers, Conference Proceedings*, London, England, 1968, vol. 183, no. 14, pp. 37-49: SAGE Publications.
- [28] R. Persky and E. Sauret, "Loss models for on and off-design performance of radial inflow turbomachinery," *Applied Thermal Engineering*, vol. 150, pp. 1066-1077, 2019.
- [29] H. E. Rohlik, "Analytical determination of radial inflow turbine design geometry for maximum efficiency," NASA1968.
- [30] I. Huntsman and H. Hodson., "An experimental assessment of the aerodynamic performance of a low-speed radial inflow turbine," presented at the 30th Joint Propulsion Conference and Exhibit, 1994.
- [31] J. D. Denton, "Some limitations of turbomachinery CFD," presented at the ASME Turbo Expo 2010: Power for Land, Sea, and Air, 2010.
- [32] C. H. Wu, "A general through flow theory of fluid flow with subsonic or supersonic velocity in turbo-machines of arbitrary hub and casing shapes," *NACA Paper*, p. 2302, 1951.
- [33] D. Japikse, "Progress in numerical turbomachinery analysis," *Journal of Fluids Engineering*, vol. 98, no. 4, pp. 592-606, 1976.
- [34] M. Casey and C. Robinson, "A new streamline curvature throughflow method for radial turbomachinery," *Journal of Turbomachinery*, vol. 132, no. 3, p. 031021, 2010.
- [35] P. Newton, T. Palenschat, R. Martinez-Botas, and M. Seiler, "Entropy generation rate in a mixed flow turbine passage," in *International Gas Turbine Congress 2015*, Tokyo, Japan, 2015.

- [36] J. D. Denton and W. N. Dawes, "Computational fluid dynamics for turbomachinery design," *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, vol. 213, no. 2, pp. 107-124, 1998.
- [37] P. G. Tucker, "Computation of unsteady turbomachinery flows: Part 1—Progress and challenges," *Progress in Aerospace Sciences*, vol. 47, no. 7, pp. 522-545, 2011.
- [38] P. G. Tucker, "Computation of unsteady turbomachinery flows: Part 2—LES and hybrids," *Progress in Aerospace Sciences*, vol. 47, no. 7, pp. 546-569, 2011.
- [39] G. M. Laskowski *et al.*, "Future directions of high fidelity cfd for aerothermal turbomachinery analysis and design," presented at the 46th AIAA Fluid Dynamics Conference, 2016.
- [40] N. Gourdain, F. Sicot, F. Duchaine, and L. Gicquel, "Large eddy simulation of flows in industrial compressors: a path from 2015 to 2035," *Philosophical Transactions of the Royal society A: Mathematical, Physical and engineering sciences*, vol. 372, no. 2022, p. 20130323, 2014.
- [41] F. Hellström, "Numerical computations of the unsteady flow in turbochargers," PhD, KTH Royal Institute of Technology, 2010.
- [42] Y. Wang, "Numerical Studies of Flow and Associated Losses in the Exhaust Port of a Diesel Engine," PhD, KTH Royal Institute of Technology, 2013.
- [43] J. Fjällman, "Large Eddy Simulations of Complex Flows in IC-Engine's Exhaust Manifold and Turbine," PhD, KTH Royal Institute of Technology, 2014.
- [44] M. G. Kofskey and C. A. Wasserbauer, "Experimental performance evaluation of a radial-inflow turbine over a range of specific speeds," in "NASA Technical Note," NASA1966.
- [45] C. Rodgers and R. Geiser, "Performance of a high-efficiency radial/axial turbine," *Journal of turbomachinery*, vol. 109, no. 2, pp. 151-154, 1987.
- [46] S. W. Spence, R. S. Rosborough, D. Artt, and G. McCullough, "A direct performance comparison of vaned and vaneless stators for radial turbines," *129*, vol. 1, pp. 53-61, 2007.
- [47] T. Cao, L. Xu, M. Yang, and R. F. Martinez-Botas, "Radial turbine rotor response to pulsating inlet flows," *Journal of Turbomachinery*, vol. 136, no. 7, p. 071003, 2014.
- [48] M. H. Padzillah, "Experimental and numerical investigation of an automotive mixed flow turbocharger turbine under pulsating flow conditions," Doctor of Philosophy (PhD), Mechanical Engineering, Imperial College London, 2014.
- [49] I. M. Khalil, W. Tabakoff, and A. Hamed, "Losses in radial inflow turbines," *Journal of Fluids Engineering*, vol. 98, no. 3, pp. 364-371, 1976.
- [50] C. Kim and K. Civinskas, "An aerodynamic analysis of a mixed flow turbine," presented at the 30th Joint Propulsion Conference and Exhibit, 1994.
- [51] P. S. Simonyi, R. J. Roelke, R. G. Stabe, B. C. Nowlin, and D. Diccico, "Aerodynamic evaluation of two compact radial-inflow turbine rotors," 1995.
- [52] N. Karamanis, R. F. Martinez-Botas, and C. C. Su, "Mixed flow turbines: Inlet and exit flow under steady and pulsating conditions," *Journal of turbomachinery*, vol. 123, no. 2, 2001.
- [53] R. Dambach, H. P. Hodson, and I. Huntsman, "An Experimental Study of Tip Clearance Flow in a Radial Inflow Turbine," *Journal of turbomachinery*, vol. 121, no. 4, pp. 644-650, 1999.
- [54] B. Lakshminarayana, *Fluid Dynamics and Heat Transfer of Turbomachinery*. John Wiley & Sons, Inc, 1996.
- [55] J. P. Johnston, "Effects of system rotation on turbulence structure: a review relevant to turbomachinery flows," *International Journal of rotating machinery*, vol. 4, no. 2, pp. 97-112, 1998.

- [56] H.-A. Schreiber, W. Steinert, and B. Küsters, "Effects of Reynolds number and free-stream turbulence on boundary layer transition in a compressor cascade," presented at the ASME Turbo Expo 2000: Power for Land, Sea, and Air, 2000.
- [57] R. J. Volino, "Separated flow measurements on a highly loaded low-pressure turbine airfoil," *Journal of Turbomachinery*, vol. 132, no. 1, p. 011007, 2010.
- [58] T. J. Praisner, E. A. Grover, M. J. Rice, and J. P. Clark, "Predicting transition in turbomachinery—Part II: Model validation and benchmarking," *Journal of Turbomachinery*, vol. 129, no. 1, pp. 14-22, 2007.
- [59] J. W. Douglas, B. S. S-M. Li, W. F. Ng, T. Sonoda, and T. Arima, "Effects of Freestream Turbulence on the Losses of a Highly-Loaded Compressor Stator Blade," presented at the ASME Turbo Expo 2003, collocated with the 2003 International Joint Power Generation Conference, 2003.
- [60] J. P. Bons, "A review of surface roughness effects in gas turbines," *Journal of turbomachinery*, vol. 132, no. 2, p. 021004, 2010.
- [61] J. H. Im, J. H. Shin, G. V. Hobson, S. J. Song, and K. T. Millsaps, "Effect of leading edge roughness and Reynolds number on compressor profile loss," presented at the ASME Turbo Expo 2013: Turbine Technical Conference and Exposition, 2013.
- [62] J. Bellucci *et al.*, "The influence of roughness on a high-pressure steam turbine stage: an experimental and numerical study," *Journal of Engineering for Gas Turbines and Power*, vol. 137, no. 1, p. 012602, 2015.
- [63] L. Xu and J. D. Denton, "The base pressure and loss of a family of four turbine blades," *Journal of turbomachinery*, vol. 110, no. 1, pp. 9-17, 1988.
- [64] G. Ciatelli and C. H. Sieverding, "The effect of vortex shedding on the unsteady pressure distribution around the trailing edge of a turbine blade," *Oceanographic Literature Review*, vol. 3, no. 45, p. 602, 1998.
- [65] C. H. Sieverding, H. Richard, and J.-M. Desse, "Turbine blade trailing edge flow characteristics at high subsonic outlet mach number," *Journal of turbomachinery*, vol. 125, no. 2, pp. 298-309, 2003.
- [66] S. Vagnoli, T. Verstraete, B. Mateos, and C. H. Sieverding, "Prediction of the unsteady turbine trailing edge wake flow characteristics and comparison with experimental data," *Proceedings of the institution of mechanical engineers, Part A: Journal of power and energy*, vol. 229, no. 5, pp. 487-497, 2015.
- [67] J. D. Coull and H. P. Hodson, "Predicting the profile loss of high-lift low pressure turbines," *Journal of Turbomachinery*, vol. 134, no. 2, p. 021002, 2012.
- [68] C. Yang, C. Ma, Y. Wang, and D. Lao, "A review of studies on turbomachinery tip gap leakage flow," *Advances In Mechanics*, vol. 31, no. 1, pp. 70-83, 2001.
- [69] A. Yamamoto, "Endwall flow/loss mechanisms in a linear turbine cascade with blade tip clearance," *Journal of Turbomachinery*, vol. 111, no. 3, pp. 264-275, 1989.
- [70] A. C. Huang, E. M. Greitzer, C. S. Tan, E. F. Clemens, S. G. Gegg, and E. R. Turner, "Blade Loading Effects on Axial Turbine Tip Leakage Vortex Dynamics and Loss," *Journal of Turbomachinery*, vol. 135, no. 5, p. 051012, 2013.
- [71] J. Gao, Q. Zheng, Tianbang Xu, and P. Dong, "Inlet conditions effect on tip leakage vortex breakdown in unshrouded axial turbines," *Energy*, vol. 91, pp. 255-263, 2015.
- [72] J. Gao, Q. Zheng, Y. Liu, and P. Dong, "Effects of blade rotation on axial turbine tip leakage vortex breakdown and loss " *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, vol. 0, no. 0, p. 0954410016656879, 2016.
- [73] M. I. Yaras and S. A. Sjolander, "Effects of Simulated Rotation on Tip Leakage in a Planar Cascade of Turbine Blades: Part I—Tip Gap Flow," *Journal of turbomachinery*, vol. 114, no. 3, pp. 652-659, 1992.

- [74] J. Tallman and B. Lakshminarayana, "Numerical simulation of tip leakage flows in axial flow turbines, with emphasis on flow physics: part II—effect of outer casing relative motion," *Journal of turbomachinery*, vol. 123, no. 2, pp. 324-333, 2001.
- [75] D. Yang, X. Yu, and Z. Feng, "Investigation of leakage flow and heat transfer in a gas turbine blade tip with emphasis on the effect of rotation," *Journal of Turbomachinery*, vol. 132, no. 4, p. 041010, 2010.
- [76] C. Zhou, H. Hodson, I. Tibbott, and M. Stokes, "Effects of Endwall Motion on the Aero-Thermal Performance of a Winglet Tip in a HP Turbine," *Journal of Turbomachinery*, vol. 134, no. 6, p. 061036, 2012.
- [77] R. Dambach, H. P. Hodson, and I. Huntsman, "Tip leakage flow: a comparison between small axial and radial turbines," in *Micro-Turbine Generators*, M. J. Moore, Ed.: Professional Engineering Publishing, 2002, pp. 97-108.
- [78] Q. Deng, J. Niu, and Z. Feng, "Study on leakage flow characteristics of radial inflow turbines at rotor tip clearance," *Science in China Series E: Technological Sciences*, vol. 51, no. 8, pp. 1125-1136, 2008.
- [79] C. De Maesschalck, S. Lavagnoli, G. Paniagua, and N. Vinha., "Aerothermodynamics of tight rotor tip clearance flows in high-speed unshrouded turbines," *Applied Thermal Engineering*, vol. 65, no. 1-2, pp. 343-351, 2014.
- [80] F. J. G. Heyes, H. P. Hodson, and G. M. Dailey, "The effect of blade tip geometry on the tip leakage flow in axial turbine cascades," *Journal of turbomachinery*, vol. 114, no. 3, pp. 643-651, 1992.
- [81] J. A. Storer and N. A. Cumpsty, "An approximate analysis and prediction method for tip clearance loss in axial compressors," *Journal of Turbomachinery*, vol. 116, no. 4, pp. 648-656, 1994.
- [82] S. Sakulkaew, C. S. Tan, E. Donahoo, C. Cornelius, and M. Montgomery, "Compressor efficiency variation with rotor tip gap from vanishing to large clearance," *Journal of Turbomachinery* vol. 135, no. 3, p. 031030, 2013.
- [83] J. D. Denton and G. Pullan, "A numerical Investigation Into The Sources of Endwall Loss in Axial Flow Turbines," in *ASME Turbo Expo 2012: Turbine Technical Conference and Exposition*, Copenhagen, Denmark, 2012, pp. 1417-1430: American Society of Mechanical Engineers.
- [84] J. Dunham, "A Review of Cascade Data on Secondary Losses in Turbines," *Journal of Mechanical Engineering Science*, vol. 12, no. 1, pp. 48-59, 1970.
- [85] C. H. Sieverding, "Recent progress in the understanding of basic aspects of secondary flows in turbine blade passages," *Journal of Engineering for Gas Turbines and Power*, vol. 107, no. 2, pp. 248-257, 1985.
- [86] O. P. Sharma and T. L. Butler, "Predictions of endwall losses and secondary flows in axial flow turbine cascades," *Journal of Turbomachinery*, vol. 109, pp. 229-236, 1987.
- [87] J. Cui, V. N. Rao, and P. G. Tucker, "Numerical investigation of secondary flows in a high-lift low pressure turbine," *International Journal of Heat and Fluid Flow*, vol. 63, pp. 149-157, 2017.
- [88] L. S. Langston, "Secondary flows in axial turbines—a review," *Annals of the New York Academy of Sciences*, vol. 934, no. 1, pp. 11-26, 2001.
- [89] N. C. Baines, "Flow development in radial turbine rotors," presented at the ASME 1996 International Gas Turbine and Aeroengine Congress and Exhibition, 1996.
- [90] L. Chen, W. Zhuge, Y. Zhang, S. Zhang, and J. Zhang, "Investigation of the secondary flow structure in the mixed flow turbine for a high pressure ratio turbocharger," presented at the ASME Turbo Expo 2008: Power for Land, Sea, and Air, 2008.

- [91] D. C. Wisler, "Loss reduction in axial-flow compressors through low-speed model testing," *Journal of engineering for gas turbines and power*, vol. 107, no. 2, pp. 354-363, 1985.
- [92] M. J. Bloxham and J. P. Bons, "Leading-Edge Endwall Suction and Midspan Blowing to Reduce Turbomachinery Losses," *Journal of Propulsion and Power*, vol. 26, no. 6, pp. 1268-1275, 2010.
- [93] Q. Deng, J. Niu, and Z. Feng, "Tip Leakage Flow in Radial Inflow Rotor of a Microturbine With Varying Blade-Shroud Clearance," in *ASME Turbo Expo 2007: Power for Land, Sea and Air*, Montreal, Canada, 2007, pp. 1081-1088: American Society of Mechanical Engineers.
- [94] Z. Schabowski and H. Hodson, "The reduction of over tip leakage loss in unshrouded axial turbines using winglets and squealers," *Journal of Turbomachinery*, vol. 136, no. 4, p. 041001, 2014.
- [95] S. W. Lee and B. J. Chae, "Effects of squealer rim height on aerodynamic losses downstream of a high-turning turbine rotor blade," *Experimental Thermal and Fluid Science*, vol. 32, no. 8, pp. 1440-1447, 2008.
- [96] N. W. Harvey and K. Ramsden, "A computational study of a novel turbine rotor partial shroud.," *Journal of turbomachinery* vol. 123, no. 3, pp. 534-543, 2001.
- [97] S. W. Lee, S. U. Kim, and K. H. Kim, "Aerodynamic performance of winglets covering the tip gap inlet in a turbine cascade," *International Journal of Heat and Fluid Flow*, vol. 34, pp. 36-46, 2012.
- [98] Z. Wei, W. Qiao, P. Shi, P. Chen, and L. Zhao, "Tip leakage flow loss reduction in a two-stage turbine using axisymmetric casing contouring," *Chinese Journal of Aeronautics*, vol. 27, no. 5, pp. 1111-1121, 2014.
- [99] B. Rosic and J. D. Denton, "Control of shroud leakage loss by reducing circumferential mixing," *Journal of Turbomachinery*, vol. 130, no. 2, p. 021010, 2008.
- [100] J. Gao, Q. Zheng, P. Dong, and X. Zha, "Control of Tip Leakage Vortex Breakdown by Tip Injection in Unshrouded Turbines," *Journal of Propulsion and Power*, vol. 30, no. 6, pp. 1510-1519, 2014.
- [101] T. Behr, A. I. Kalfas, and R. S. Abhari, "Control of rotor tip leakage through cooling injection from the casing in a high-work turbine," *Journal of Turbomachinery*, vol. 130, no. 3, p. 031014, 2008.
- [102] Y. Wang, Y. Song, Jianyang Yu, and F. Chen, "Effect of cooling injection on the leakage flow of a turbine cascade with honeycomb tip," *Applied Thermal Engineering*, vol. 133, pp. 690-703, 2018.
- [103] L. Qi and Y. Zhou, "Turbine Blade Tip Leakage Flow Control by Unsteady Periodic Wakes of Upstream Blade Row," *Procedia Engineering* vol. 80, pp. 202-215, 2014.
- [104] J. D. Denton and L. Xu, "The exploitation of three-dimensional flow in turbomachinery design," *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, vol. 213, no. 2, pp. 125-137, 1998.
- [105] H. D. Weingold, R. J. Neubert, R. F. Behlke, and G. E. Potter, "Bowed stators: an example of CFD applied to improve multistage compressor efficiency," *Journal of Turbomachinery*, vol. 119, no. 2, pp. 161-168, 1997.
- [106] A. Duden, I. Raab, and L. Fottner, "Controlling the Secondary Flow in a Turbine Cascade by Three-Dimensional Airfoil Design and Endwall Contouring," *Journal of Turbomachinery*, vol. 121, no. 2, pp. 191-199, 1999.
- [107] M. W. Benner, S. A. Sjolander, and S. H. Moustapha, "An empirical prediction method for secondary losses in turbines—part II: a new secondary loss correlation," *Journal of Turbomachinery*, vol. 128, no. 2, pp. 281-291, 2006.

- [108] T. J. Praisner *et al.*, "Toward the Expansion of Low-Pressure-Turbine Airfoil Design Space," *Journal of Turbomachinery*, vol. 135, no. 6, p. 061007, 2013.
- [109] H. Sauer, R. Muller, and K. Vogeler, "Reduction of Secondary Flow Losses in Turbine Cascades by Leading Edge Modifications at the Endwall," *Journal of Turbomachinery*, vol. 123, no. 2, pp. 207-213, 2001.
- [110] Ö. H. Turgut and C. Camcı, "A nonaxisymmetric endwall design approach and its computational assessment in the NGV of an HP turbine stage," *Aerospace Science and Technology*, vol. 47, pp. 456-466, 2015.
- [111] S. Shahpar, S. Caloni, and L. d. Prieelle, "Automatic design optimization of profiled endwalls including real geometrical effects to minimize turbine secondary flows," *Journal of Turbomachinery*, vol. 139, no. 7, p. 071010, 2017.
- [112] S. I. Benton, J. P. Bons, and R. Sondergaard, "Secondary Flow Loss Reduction Through Blowing for a High-Lift Front-Loaded Low Pressure Turbine Cascade," *Journal of Turbomachinery*, vol. 135, no. 2, p. 021020, 2013.
- [113] T. Kawai, S. Shinoki, and T. Adachi, "Secondary flow control and loss reduction in a turbine cascade using endwall fences," *JSME international journal. Ser. 2, Fluids engineering, heat transfer, power, combustion, thermophysical properties*, vol. 32, no. 3, pp. 375-387, 1989.
- [114] J. Gao and Q. Zheng, "Comparative Investigation of Unsteady Flow Interactions in Endwall Regions of Shrouded and Unshrouded Turbines," *Computers and Fluids*, vol. 105, pp. 204-217, 2014.

Chapter 3: Methodology

This chapter describes the research methodology adopted for the current thesis. An overview of the multi-injection turbine housing concept is explained. Details of the turbine rotor and stator being investigated is also presented. This is followed by detailed methodology of the CFD simulations, optimization method, prototype building, and the experimental testing. A method of calculating turbine performance parameters with injection, as well as a method of evaluating entropy generation within the rotor passage is also presented.

3.1 Overview of Multi-Injection Turbine Housing

A novel method designed to use forced secondary flows to alter the flow within a mixed flow turbine was proposed, which aims to minimize the losses occurred within the rotor passage thereby increasing its power and efficiency. The secondary injection system could be integrated to increase the turbocharger performance once design optimization and other techniques have been applied, providing further improvements in performance. The turbine casing was modified in the shroud area through which secondary flow is injected.

The secondary injection system consists of an injection slot which extends circumferentially along the entire shroud, as shown in Figure 3.1. The width and location of the injection slot can be finalized through optimization. Air is injected through the injection slot, which aims to impart momentum to the main passage flow in the rotor, improving the flow field in the rotor passage. At an optimum injection speed and direction, it is hypothesized that the injected flow can result in lower aerodynamic losses within the rotor, thereby improving rotor efficiency. A secondary objective of the injection is also to increase the work extracted by the rotor blades.

Figure 3.1: Schematic of turbine geometry with secondary injection slot

Five injection parameters are varied for this study. These are: injection slot location, injection slot width, injection flow velocity, injection yaw angle θ , and injection pitch angle α . The injection slot location is defined by the axial distance between the blade tip leading edge and the start of the injection slot, while the injection slot width is the axial distance between the start and end of the injection slot. The injection yaw angle θ is defined as the circumferential angle measured on the z - θ plane, with a positive yaw angle in the same direction as the rotation of the turbine, and the pitch angle α is defined

as the radial angle made between the injected flow and the z-r plane. The definitions of the injection parameters are visualized in Figure 3.2.

Figure 3.2: Definition of Injection Parameters

The overall methodology adopted is shown in Figure 3.3. Single passage design and optimization of the injection was first conducted. Using the results from the single passage simulations, a prototype was manufactured and tested experimentally. A full stage CFD model was then built to validate CFD results against the experimental data, and from there further simulations could be conducted on additional cases. Using data obtained from the full stage simulations, a re-optimization could then be conducted to obtain the best injection parameters for the turbine studied. For the current study, the source of air for the secondary flow injection is from an additional air source, but future studies will explore the usage of part of the engine exhaust gas as an injection source.

Figure 3.3: Overall Research Methodology Adopted

3.2 Turbine Investigated

The turbine being studied is a mixed flow turbine dubbed ‘Rotor A’ used in automotive turbocharger systems, designed in Imperial College London by Abidat [1]. The turbine

consists of 12 blades, and has leading edge and trailing edge tip diameter of 95.14mm and 78.65mm respectively. It is designed with a constant inlet blade angle of 20° . Details of the turbine geometry is summarized in Table 3.1, and further details can be found in [1]. A 3D model of the turbine rotor is shown in Figure 3.4.

Figure 3.4: 3D Model of Rotor A

Table 3.1: Rotor A Geometric Parameters

Leading edge tip diameter (mm)	95.14
Trailing edge tip diameter (mm)	78.65
Leading edge span height (mm)	18.00
Trailing edge span height (mm)	25.79
Cone angle ($^\circ$)	40
Leading edge blade angle ($^\circ$)	20
Length of axial chord (mm)	40
Number of blades	12
Tip gap height (% of blade span)	3

The volute and nozzle vanes used for the current investigation was originally designed and used by Rajoo [2] for investigating a variable geometry turbine system using pivoting vanes. The volute is shown in Figure 3.5, and was designed to have an exit flow angle of 69° with respect to the radial direction.

Figure 3.5: Schematic of Volute [2]

The nozzle vanes used is a lean vane, shown in Figure 3.6. This design was found by Rajoo to have a higher swallowing capacity and lower efficiency drop at off-design operating points compared to a traditional straight vane design [2]. The number of vanes in the nozzle ring is 15 as was in the original design. Rajoo originally designed the nozzle vanes with a pivoting mechanism as part of a VGT, thus there was a clearance of 0.15mm at both endwalls. For the current study, however, the vane angle is fixed at 65° , and the vane shroud wall clearance is reduced to 0° since it is now part of the nozzle ring assembly, while the hub wall clearance is increased to 0.3mm to maintain the same vane height.

Figure 3.6: Schematics of Nozzle Vane [2]

3.3 Simulation Methodology

The commercial software ANSYS CFX 18.2 was used for the numerical simulations of the turbine. Geometry preparation was done using SolidWorks software, while meshing

was generally done using ANSYS ICEM CFD. Steady-state CFD simulations were conducted, and the k-epsilon turbulence model with scalable wall functions was used for all computations, as it is widely used in turbomachinery simulations and is able to produce reasonable accuracy in turbomachinery applications. The other reason for using the k-epsilon model is that the y^+ values for the currently used mesh was generally in the order of 10^1 , and the usage of denser meshes to have a y^+ value of less than 1 was prohibited by the limited computing resources available, especially for optimization which requires many hundreds of simulations to be run. Thus, the k-omega based models were not used. The k-epsilon model was found to be able to predict bulk flow parameters well, and is also able to capture important flow features in a mixed flow turbine [3]. The scalable wall function in CFX virtually moves all mesh points at a y^+ value of less than 11.06 to a y^+ value of 11.06, so that the first mesh point is always outside the viscous sublayer [4]. This avoids inconsistencies in results faced with fine meshes when using wall functions, and allows for fairly mesh-independent results. A convergence criteria of $10e^{-5}$ was used for all simulations to ensure sufficient convergence of the solutions.

3.3.1 Mesh Independence Study

To make sure that the mesh density employed is sufficient, a mesh independence study was first conducted using single passage baseline turbine simulations. Two different mesh densities were tested: 418,272 nodes (Mesh 1) and 561,771 nodes (Mesh 2). The mesh density was increased mainly by refining the mesh near the turbine blade and the tip gap. The simulations were conducted at 30,000rpm over a range of operating conditions. Boundary conditions used are those stated in Chapter 3.3.2 (see Table 3.2 for summary), which will be elaborated in Chapter 3.3.2.

Figure 3.7 shows the results of the mesh independence study, presented in the form of the performance parameters mentioned in Chapter 3.7. The results show that there is negligible difference in the turbine performance parameters between the two mesh densities used, and thus a mesh density of at least 420,000 nodes per rotor passage is recommended for subsequent simulations. This mesh density is also consistent with results from Newton et al. [3]

Figure 3.7: Mesh Independence Results

3.3.2 Single Passage Simulations

For the single passage simulations, only a single rotor passage domain was included in the simulations. The injection slot was created as an additional surface on the shroud, as shown in Figure 3.8. This will be known as the secondary inlet. The inlet of the rotor domain was also extended to allow for the inlet flow to develop before reaching the rotor. A structured block meshing method was used for the simulation domain, with hexagonal mesh elements used, with an O-grid created around the rotor blade. The rotor mesh consists of 505628 nodes. The wall y^+ was below 50 for all simulation cases, meaning that the wall function approach in the k-epsilon model is valid. 20 nodes were

also included in the tip gap region to ensure that the flow in this area is sufficiently resolved. This is especially important as the injected flow is in this region and sufficient mesh density is needed for adequate resolution of the injected flow and its interactions with the main passage flow. The single passage mesh is shown in Figure 3.9.

Figure 3.8: Secondary injection single passage domain

Figure 3.9: Single Passage CFD Mesh

At the inlet of the domain, a mass flow boundary condition, together with an inlet total temperature of 333K was imposed. The flow direction of the inlet flow was prescribed based on the exit flow angle of the volute. At the outlet, a static pressure of 1atm was prescribed. A periodic boundary was set up at the side walls with rotational periodicity, and for the blade, hub and shroud walls a smooth, adiabatic wall boundary condition was prescribed. A velocity inlet was imposed as the boundary condition at the secondary inlet for injection simulations, while for baseline simulations a wall boundary condition similar to the rest of the shroud was used. The injected flow has a total temperature of 333K, similar to the inlet total temperature of the main passage flow. A summary of the boundary conditions used can be found in Table 3.2.

Table 3.2: Single Passage CFD Boundary Conditions

Domain	Boundary Condition
Primary inlet (main passage inlet)	Stationary inlet with mass flow, total temperature 333K, flow direction based on volute exit flow
Primary outlet (main passage outlet)	Stationary outlet with 1atm static pressure
Hub	Wall, smooth, adiabatic, rotating
Shroud	Wall, smooth, adiabatic, counter-rotating
Blade	Wall, smooth, adiabatic, rotating
Side walls of passage	Rotational periodicity
Secondary inlet (injection inlet)	Stationary inlet with velocity, total temperature 333K, flow direction prescribed
Secondary inlet (baseline)	Wall, smooth, adiabatic, counter-rotating

For the injection simulations, the injection parameters, listed in Table 3.3 were varied. Ideally the parameters should be allowed to vary without any constraint to explore the full range of design space. However, this is not possible as problems with meshing and numerical problems with the solver will arise when the parameters exceed a certain range. To prevent problems with automatic meshing in optimization, the slot location is limited to between 5.5mm to 16.5mm, and the slot width to vary between 1mm to 5mm. This is shown in Figure 3.10, where the region shaded in blue represents the region where the slot can exist, with the diagram showing a slot of width 5mm. This

region covers about two-thirds of the entire chord of the blade, ensuring that a sufficient range of design can be achieved. Lower and upper limits are placed on the injection velocity, yaw angle and pitch angle as well, to ensure that the CFD solver does not crash and the parameters have realistic values. The range of parameters possible can be found in Table 3.3.

Figure 3.10: Limits of Injection Location and Width

Table 3.3: Injection Parameter and Ranges

Parameter	Description	Lower Limit	Upper Limit
x1	Slot Location (from blade tip LE)	5.5mm	16.5mm
x2	Slot Width	1mm	5mm
x3	Injection Velocity	10m/s	260m/s
x4	Injection Yaw Angle	-80°	80°
x5	Injection Pitch Angle	10°	89°

The single passage simulations were run at 30,000rpm to simulate engine part load conditions. 8 operating points, chosen to represent the turbine flow over a wide range of operating conditions, were simulated to create a performance speed line as shown in Table 3.4.

The turbine inlet and outlet parameters (e.g. temperature and pressure) were taken at two user-defined surfaces located upstream and downstream of the rotor blade respectively, shown as the black surfaces in Figure 3.11.

Table 3.4: Turbine Operating Points, 30,000rpm

Point	Mass flow (kg/s)
1	0.13015
2	0.16375
3	0.19887
4	0.23262
5	0.26012
6	0.31115
7	0.36284
8	0.39379

Figure 3.11: User-defined Surfaces for Turbine Inlet and Outlet Measurements

3.3.3 Full Turbine Stage Simulations

For the full stage turbine simulations, the entire turbine stage, which includes the volute, nozzle, rotor, as well as the injection duct was included in the simulations. The meshing for each part was done separately, and the final computational mesh was put together in ANSYS CFX. For the rotor and nozzle vanes, a structured blocked meshing method similar to the single passage simulations was used, while for the main volute and injection volutes, an unstructured tetrahedral mesh with inflation layers is used. An extended inlet duct was also added to both the main and injection inlet domains to allow

for the flow to develop before reaching the volute. The rotor domain has a mesh density of 535436 nodes per passage, and the entire simulation domain contains about 9 million nodes, with the breakdown provided in Table 3.5. The complete simulation domain (domain (A)) can be seen in Figure 3.12, with the injection duct highlighted in blue. An alternative simulation domain without the injection duct was also used, shown on the right as domain (B) in Figure 3.12.

Table 3.5: Full Stage CFD Mesh Statistics

Region	Mesh size (nodes)
Inlet duct	242,320
Volute	730,016
Nozzle vanes	1,272,000
Rotor	6,425,232
Injection duct	347,462
Total	9,017,030

Figure 3.12: Full stage turbine CFD simulation domain with injection: left) with injection duct, right) without injection duct

Mass flow boundary condition of 0.033kg/s was used for the injection inlet duct, whilst a total pressure boundary condition was used for the main inlet duct. For domain (B), a velocity inlet was specified for the secondary inlet (rotor injection slot) boundary condition. Stage (mixing plane) interface was used between the nozzle vanes and the rotor inlet to simulate the average flow characteristics, while frozen rotor interface was used between the injection duct and the rotor injection slot due to convergence issues. All wall boundary conditions were adiabatic and smooth, except for the injection duct, vane blades and vane shroud. These surfaces were prescribed a surface roughness of

8.8×10^{-4} mm equivalent roughness, as this was the actual roughness measured on the parts. A fillet of radius 0.5 mm was also added to CFD model of the base of the nozzle vanes to match imperfections in the manufactured nozzle ring. A summary of the boundary conditions used can be found in Table 3.6.

Table 3.6: Full Stage CFD Boundary Conditions

Domain	Boundary Condition
Primary inlet (of main inlet duct)	Stationary inlet with total pressure, total temperature 333K, flow normal to inlet
Primary outlet (rotor outlet)	Stationary outlet with 1 atm static pressure
Rotor hub	Wall, smooth, adiabatic, rotating
Rotor shroud	Wall, smooth, adiabatic, counter-rotating
Rotor blade	Wall, smooth, adiabatic, rotating
Nozzle vane shroud	Wall, 8.8×10^{-4} mm equivalent roughness, adiabatic, stationary
Nozzle vane blade	Wall, 8.8×10^{-4} mm equivalent roughness, adiabatic, stationary
Walls of injection duct	Wall, 8.8×10^{-4} mm equivalent roughness, adiabatic, stationary
All other walls	Wall, smooth, adiabatic, stationary
Secondary inlet (injection duct)	Stationary inlet with mass flow, total temperature 333K, flow normal to inlet
Secondary inlet (no injection duct)	Stationary inlet with specified velocity, flow direction and total temperature 333K
Nozzle vanes to rotor interface	Stage (mixing plane)
Injection duct to rotor interface	Frozen rotor
All other interfaces	General connection

The simulations were conducted at a rotational speed of 30,000 rpm and 48,000 rpm, which was 50% and 80% of the design speed of the turbine respectively, in order to validate the CFD results against experimental data at a range of operating conditions. 8 operating points were run for each operating speed, shown in Table 3.7 below. Similar to the single passage simulations, a user-defined surface (shown as the black surface in

Figure 3.13) was created to measure the turbine outlet flow conditions, while the turbine inlet flow conditions were measured at the inlet of the rotor domain (shown as the green surface in Figure 3.13) to simulate the effect of measuring the flow at the measuring plane in experiments.

Table 3.7: Turbine Operating Points at 30,000rpm and 48,000rpm

Point	Inlet Total Pressure (Pa)	
	30,000rpm	48,000rpm
2	116894	129521
3	121123	137569
4	125840	148994
5	130114	154810
6	138934	163455
7	149343	173394
8	156013	187047
9	165167	206935

Figure 3.13: Full Stage User-defined Surfaces for Turbine Inlet and Outlet Measurements

Besides validation against experimental data, some additional simulations were also run at 30,000rpm to assess potential performance improvement scenarios through flow injection. The single passage optimization at 30,000rpm resulted in an overall injection mass flow rate of about 0.04kg/s. However, due to an error in the measurement of the injection mass flow rate during experimental testing, the actual injection mass flow in the experiments was only 0.033kg/s. Thus, during CFD validation against experimental data this lower value is used. A set of full stage simulations with the intended injection mass flow of 0.04kg/s was also run. To assess the potential benefits of secondary flow injection, simulations without the nozzle vanes, as well as simulations with injection flow as a velocity boundary condition over the shroud (domain (B) in Figure 3.12) were also run. Injection flow as a boundary condition signifies a perfect injection condition, and only one rotor passage was included in the simulations with periodic boundary conditions used. This was done to save computational cost without sacrificing accuracy of results. The vaneless simulation domain is the same as the vaned one, the only exception being the lack of inlet nozzle vanes. While this set up results in an unusually large gap between the volute and the rotor blade inlet, to decrease the size of the gap the entire volute had to be redesigned, which was not possible at the current moment. Nevertheless, the vaneless set up can provide an idea of the effectiveness of injection in vaneless turbines. Figure 3.14 and Table 3.8 show a summary of the type of simulations run.

3.4 Optimization Methodology

In order to find out the best set of parameters to be used for injection, an optimization was done using genetic algorithm. The genetic algorithm is an evolutionary algorithm that seeks to mimic biological evolution through natural selection. The algorithm starts with an initial population of individual solutions, where the parameter values are randomly determined. A fitness function is defined based on the optimization goals. The fitness of the individuals are then evaluated using the fitness function, and selected individuals, usually those of better fitness, are then chosen as ‘parents’ to produce ‘offspring’ to create the next generation of individuals. Offspring can be produced either through crossover (exchange of parameters) of two parents, or mutation (randomly changing parameters) of a single parent, illustrated in Figure 3.15. After multiple generations, the design space will be populated by solutions with different

combinations of parameter values, and the population will ‘evolve’ towards the optimum solution. This method is effective in optimization of problems that are discontinuous, non-differentiable, stochastic, or highly nonlinear [5], and has been shown to be effective in optimizing the geometry of a turbocharger turbine wheel [6]. However, the main drawback of genetic algorithm is that it is sometimes inefficient in finding the true minimum value of the fitness function.

Figure 3.14: Full Stage Injection CFD Cases

The optimization was conducted at a single operating point at 50% turbine design speed (30,000rpm point 5 in Table 3.4). This point was chosen because it is close to the peak efficiency at 30,000rpm. Therefore, the effect of shifting of the operating point due to additional mass is removed, and any gain in efficiency must be due to flow field improvement and reduction of losses.

Table 3.8: Description of Full Stage Injection CFD Cases

Simulation Case	Remarks
A	Injection mass flow 0.033kg/s with injection duct (domain (A)), vaned, for validation
B	Injection mass flow 0.033kg/s, no injection duct (domain (B)), vaned, velocity inlet specified for injection
C	Injection mass flow 0.04kg/s with injection duct (domain (A)), vaned
D	Injection mass flow 0.04kg/s, no injection duct (domain (B)), vaned, velocity inlet specified for injection
E	Injection mass flow 0.04kg/s with injection duct (domain (A)), vaneless
F	Injection mass flow 0.04kg/s, no injection duct (domain (B)), vaneless, velocity inlet specified for injection

Figure 3.15: Crossover and Mutation in Genetic Algorithm

The genetic algorithm in MATLAB's optimization toolbox was used for the optimization. The five parameters (x1 to x5) shown in Table 3.3 were optimized within their respective bounds. The objective of the optimization is to maximize efficiency, or to minimize the following fitness function:

$$1 - \eta_{TS} \quad (3-1)$$

An overview of the workflow for the current implementation of the optimization is presented in Figure 3.16. ANSYS CFX was coupled into the genetic algorithm to calculate the fitness function. This was done through a user-written MATLAB function. The user-written function returns the fitness value for a certain set of parameter values which is used in genetic algorithm. For each generation, a population size of 10 was chosen, and the elite count was set to be 1 to ensure that the individual with the lowest fitness function value is carried forward to the next generation. To fully explore the design space and to prevent results from falling into local minima, a crossover fraction of 0.6 was set, which meant that random values for the parameters will be generated in 4 out of the 10 individuals in a population (these are known as mutation children). The remaining 5 individuals are crossover children whose parameter values are a combination of 2 parents from the previous generation, selected based on their fitness value. The optimization was performed until the average change of the best fitness function value is less than 10^{-6} for 8 generations (stall generations). Table 3.9 shows a summary of the options used in the Genetic Algorithm. These options were chosen based on experience which provided a good convergence rate of the results towards the optimized value, with the final optimization achieved over 87 generations (870 simulations). The set of parameters giving the overall lowest fitness value was chosen as the optimized point.

Figure 3.16: General workflow for the current optimization process

Table 3.9: Settings for Genetic Algorithm

Population size	10
Stall Generations	8
Crossover children	5
Mutation children	4
Elite children	1
No. of generations before final result	87

3.5 Design of Duct for Injection

Once the single passage optimizations provided a set of solutions for the injection parameters, a prototype to experimentally test the multi-injection turbine housing could be built. The first step was to devise a way for flow to be injected over the shroud through a circumferential slot in the optimized direction obtained through CFD simulations. In order to achieve this, a small volute is designed to direct the flow to the injection slot.

The basic method for volute design for a radial turbine described in [7] was adapted and used here. To start the design process, the desired exit flow angle α_1 , which in this case is the injection yaw angle, was used as an input. In addition, the volute exit radius r_1 and area A_1 were already known from the location and size of injection slot. Assuming a free vortex flow through the volute, the circumferential velocity can be expressed as:

$$rC_\theta = \text{constant} = K \quad (3-2)$$

And the continuity equation states that:

$$m_\theta = \rho_\theta A_\theta C_\theta \quad (3-3)$$

At the volute exit, the exit flow angle is given by:

$$\tan \alpha_1 = \frac{C_{\theta 1}}{C_{m1}} \quad (3-4)$$

where C_{m1} is the exit meridional flow velocity. From equation 3-2:

$$C_{\theta 1} = C_{\theta 0} \frac{r_0}{r_1} = C_0 \frac{r_0}{r_1} \quad (3-5)$$

And from the equation 3-3:

$$C_{m1} = \frac{\rho_0 A_0}{\rho_1 A_1} C_0 \quad (3-6)$$

Combining the three equations 3-4, 3-5 and 3-6, a relationship between the exit flow angle and the inlet and outlet area to radius ratios can be obtained:

$$\tan \alpha_1 = \frac{\rho_1 A_1 / r_1}{\rho_0 A_0 / r_0} \quad (3-7)$$

Since A_1 , r_1 , and α_1 were predetermined, and the fluid densities were known as well, the inlet area to radius ratio A_0/r_0 could be determined. For the current case, the inlet radius is constrained by space around the rotor, therefore an inlet radius was set as well. Therefore, the inlet area could also be determined.

Assuming a uniform distribution of mass flow around the volute:

$$m_\theta = m \left(1 - \frac{\theta}{2\pi}\right) \quad (3-8)$$

Combining the equation 3-8 with the equation 3-3 and 3-2:

$$\frac{A_\theta}{r_\theta} = \frac{m}{\rho_\theta K} \left(1 - \frac{\theta}{2\pi}\right) \quad (3-9)$$

where r_θ is the radius of the centroid of the section at an azimuth angle θ . Thus, as long as density variations are negligible, the ratio A_θ/r_θ must vary linearly with the azimuth angle θ to achieve a uniform mass flow distribution around the volute. Using the known inlet conditions (subscript 0), the constant K could then be determined. Therefore, the area to radius ratio at each azimuth angle could be determined, and appropriate areas and centroid radii could be chosen for the volute design. Figure 3.17 shows the final designed area and radius of the volute at different azimuth angles.

To complete the design, an appropriate cross section was selected for the volute which satisfies the area and centroid radius computed above at each azimuth angle. The final design can be shown here in Figure 3.18.

Figure 3.17: Area and radius distribution of designed injection volute

Figure 3.18: Final design of injection volute

The finalized volute is designed to direct the flow at an injection yaw angle of 80° and a pitch angle of 60° based on the single passage optimization results. The finalized designed was then made into a prototype using laser sintering additive manufacturing technique. This is shown in Figure 3.19, with the injection slot sealed up for baseline turbine testing.

Figure 3.19: Manufactured injection duct

3.6 Experimental Methodology

The designed prototype is tested experimentally at the Imperial College London Turbocharger Test Rig. The test rig is a cold-flow turbine testing facility which the turbine is loaded with an eddy-current dynamometer. This means that the turbine power test range is not limited by the performance limit of compressors and this allows for turbine performance maps that are much wider than traditional test rigs that are loaded by a compressor. The test rig is able to test the turbine in both steady and pulsating flow conditions, but for this investigation only steady flow testing was conducted. Figure 3.20 shows a schematic of the turbocharger test rig.

Air is supplied via 3 Ingersoll Rand screw type compressors capable of supplying a maximum flow rate of 1.2kg/s at 5 bar absolute pressure. The air mass flow rate can be controlled through a 4 inch electrically actuated main control valve, with a 1 inch electrically actuated bypass control valve available as well for finer control. The air is then heated by a 72kW electric heater to around 330K to 350K to prevent condensation from occurring during turbine expansion. From there, the flow splits into two individual limbs, termed the inner limb and outer limb. The limbs are set up for testing of twin and double entry turbines, each with their own electrically actuated control valves. For the current testing both limb control valves were left fully open. The flow mass flow was measured at each individual limb using a McCrometer v-cone flow meter. After that, the air is directed through a pulse generator which can generate pulsating flow for the

turbine. The pulse generator was not used for the current testing. A spring loaded guillotine safety valve is located downstream of the pulse generator. The guillotine valve was armed prior to each round of testing, and will immediately cut off air supply if any of the pre-defined safety limits were exceeded. Further downstream of the guillotine valve is the measurement plane, where the pressure sensors and thermocouples that measure the turbine inlet conditions are located. A converging duct is located after the measurement plane, which merges the flow from the two limbs together into a single passage. This is joined to the turbine volute.

Figure 3.20: Schematic of turbocharger test rig [8]

The turbine is loaded by an eddy current dynamometer. The eddy current dynamometer is water cooled to dissipate the heat created from extracting turbine power, and is capable of extracting a maximum power of 60kW. Turbine torque can be directly

measured by the load cell on the dynamometer, and an optical speed sensor is also located on the dynamometer to measure the turbine speed. As mentioned previously, the dynamometer circumvents the problem of narrow operating range of compressors, and allows for far wider turbine performance maps to be generated.

For the injection system, air is fed to the injection volute through a tube which diverts heated air from the main air supply, as shown below in Figure 3.21. An electrically actuated control valve is used to control the mass flow through the tube. The mass flow of the injected flow is measured using a nozzle flow meter. A pressure sensor is installed at the inlet of the injection duct, while a temperature sensor is installed at the outlet of the nozzle flow meter, in order to measure the pressure and temperature of the injected fluid respectively.

Figure 3.21: Experimental set up with injection

3.6.1 Mass Flow Measurement

Two different mass flow meters were used to measure the main turbine flow and the secondary injection flow rates. For the main turbine flow, the overall mass flow rate is the sum of the mass flow rates at each individual limb. The mass flow rates at the two limbs are each measured by a McCrometer v-cone flow meter, shown in Figure 3.22. The v-cone flow meter is a differential pressure flow meter, similar to orifice plate and nozzle flow meters. However, the main advantages of the v-cone are that it can generate useful differential pressures at lower flow rates than nozzles or orifice plates, and it can provide more accurate flow measurement under non-ideal flow profiles [9]. The v-cone

flow meter requires the measurement of the differential pressure between the high and low tapings as well as the gas temperature.

Figure 3.22: Schematics of the V-Cone Flow Meter

The equation used to calculate the flow mass flow rate is given by:

$$\dot{m} = \frac{\pi}{4} \sqrt{2\rho} \frac{D^2 \beta^2}{\sqrt{1 - \beta^4}} C_d F_a Y \sqrt{\Delta P} \quad (3-10)$$

where ρ is the density, D is the inner diameter of the pipe, β is the meter beta ratio, C_d is the discharge coefficient, F_a is the material thermal expansion factor, Y is the gas expansion factor, and ΔP is the differential pressure across the flow meter.

The beta ratio β is given by:

$$\beta = \sqrt{1 - \frac{d^2}{D^2}} \quad (3-11)$$

where d is the outer diameter of the cone, and D is the inner diameter of the pipe.

The gas expansion factor Y is given by:

$$Y = 1 - (0.649 + 0.696\beta^4) \frac{U_1 \Delta P}{\gamma P} \quad (3-12)$$

where U_1 is a unit conversion factor, P is the operating pressure, and γ is the gas isentropic exponent.

The material thermal expansion factor F_a is dependent on the thermal expansion coefficient of the v-cone and the temperature, but under normal operating conditions the deviation is less than 0.1% from unity, and hence its value can be assumed as 1. The

discharge coefficient C_d is a function of the Reynolds number, but over a large range of Reynolds numbers (17,000 to 430,000) the value is constant to four significant figures. The range translates to a mass flow rate of 20g/s to 500g/s in each limb, which essentially covers all the test conditions. Thus, constant values of 0.8385 and 0.8472 was used for the two v-cones.

The injection mass flow rate was measured using a nozzle flow meter designed and built according to ISO 5167 standards [10]. The nozzle is a long radius nozzle with an inlet inner diameter of 54.1mm, and an outlet inner diameter of 26.6mm, as shown in Figure 3.23. Pressure tapings were located 54.1mm upstream and 27.05mm downstream of the nozzle inlet face in accordance to the ISO standards. A thermocouple was also located downstream of the nozzle for temperature measurements. Straight pipe of sufficient length was catered upstream and downstream of the nozzle in accordance to the standards as well to ensure a well developed velocity profile for accurate mass flow measurements.

Figure 3.23: Schematic of Nozzle Flow Meter

For the nozzle flow meter, the equation for mass flow rate is given by:

$$\dot{m} = \frac{\pi}{4} \sqrt{2\rho} \frac{d}{\sqrt{1-\beta^4}} C_d Y \sqrt{\Delta P} \quad (3-13)$$

where d is the nozzle outlet inner diameter, which is 26.6mm for the current nozzle.

The beta ratio for the nozzle flow meter is given by:

$$\beta = \frac{d}{D} \quad (3-14)$$

where D is the nozzle inlet inner diameter.

The discharge coefficient C_d is given by:

$$C_d = 0.9965 - 0.00653 \sqrt{\frac{10^6 \beta}{Re_D}} \quad (3-15)$$

where Re_D is the upstream pipe Reynolds number. The uncertainty in C_d is 2.5% for the current design.

The gas expansion factor Y is given by:

$$Y = \sqrt{\left(\frac{\gamma \Pi^{\frac{2}{\gamma}}}{\gamma - 1}\right) \left(\frac{1 - \beta^4}{1 - \beta^4 \Pi^{\frac{2}{\gamma}}}\right) \left(\frac{1 - \Pi^{\frac{\gamma-1}{\gamma}}}{1 - \Pi}\right)} \quad (3-16)$$

where $\Pi = P_{down}/P_{up}$, the downstream to upstream pressure ratio. The uncertainty in Y is $2 \frac{\Delta P}{P_1} \%$.

3.6.2 Pressure Measurement

The pressure measurements were made through static-hole pressure tappings perpendicular to the flow direction. The tappings are located at the v-cone flow meters, the measurement plane, the nozzle flow meter, the turbine exit duct, as well as the inlet to the injection duct. A pneumatic tube connects each pressure tapping to a 24 channel Scanivalve system, which contains two high-accuracy strain gauge pressure transducers. A Druck PDCR 22 was used for low pressure (<0.35bar) measurements, while a Druck PDCR 23D was used for high pressure (<3.5bar) measurements. Each Scanivalve channel consists of two ports, in which both ports can be used to output a differential pressure, or the low port can be left open to measure the pressure relative to atmospheric pressure. The atmospheric pressure was recorded by a dial barometer so the absolute pressure could be obtained.

3.6.3 Temperature Measurement

Thermocouples were located at various locations of the test rig for measurement of temperature for experimental purposes and also to monitor the rig for safety purposes.

For measurement purposes, an E-type thermocouple is located at each v-cone for mass flow measurements, and one T-type thermocouple is located at each limb at the measurement plane for turbine inlet temperature measurements. A K-type thermocouple is located at the exit of the nozzle flow meter for mass flow measurements of the nozzle flow meter, and also serves a dual purpose in measuring the injection flow temperature.

The other thermocouples were used mainly for monitoring and safety purposes. The temperature at the heater, bearings, dynamometer stator plates and dynamometer cooling water were all monitored.

The temperature measurement of a moving fluid requires a correction due to compressibility effects. Due to compressibility, the measured temperature will be between the static and stagnation temperature of the fluid. Thus, a recovery factor is introduced to correct the temperature:

$$r = \frac{T_{meas} - T_s}{T_0 - T_s} \quad (3-17)$$

where T_{meas} is the measured temperature, T_s is the actual static temperature, and T_0 is the actual stagnation temperature. The actual static temperature is then obtained through the following expression:

$$T_s = \frac{T_{meas}}{1 + r \left(\frac{\gamma - 1}{2} \right) Ma^2} \quad (3-18)$$

where γ is the gas isentropic exponent, and Ma is the Mach number. In the experiments, the values of r , T_0 , T_s and Ma were computed through an iterative process.

3.6.4 Turbine Speed Measurement

The turbine speed is measured using an infra-red optical sensor (Omron EE-SX1103) triggered by a 20-toothed encoder mounted on the turbine shaft, shown in Figure 3.24. The 20 pulses per revolution are then converted into a single pulse and sent to a 16-bit counter module which counts the number of pulses from a 16MHz clock. An output voltage proportional to the number of clock pulses recorded per unit time (inversely proportional to the turbine speed) is produced. This voltage is read directly by the analogue input Fieldpoint module, FP-AI-110 and recorded. From this, the turbine speed can be precisely measured.

Figure 3.24: Turbine Speed Measurement [11]

3.6.5 Torque and Power Measurement

The torque produced by the turbine was measured through the reaction of the eddy current dynamometer, using an Interface Miniature Beam force transducer employed as the load cell. The output signal of the load cell was logged for the entire duration of a test point, and also after the turbine is brought to rest. The output signal after the turbine is brought to rest is used as the zero offset value. The torque of the turbine is then the difference between the torque signal during test and the zero offset.

With the turbine torque, the turbine power output can then be calculated using the turbine speed:

$$W_{act} = 2\pi N\tau \quad (3-19)$$

where N is the turbine speed and τ is the turbine torque.

3.6.6 Uncertainty of Measurements

The uncertainty values of the measurements were calculated using a statistical method method presented in [12]. This method was also used to calculate the uncertainties of the measurements by previous experimentalists using the turbocharger test rig (see [11, 13]).

In this thesis, an uncertainty value of twice the standard deviation $\pm 2\sigma$ was given for the measured values, giving a confidence level of 95.4%, assuming a normal distribution for the measurements. The standard deviation σ of a parameter $y =$ is given by:

$$\sigma = \sqrt{\frac{1}{n-v} \sum_{i=1}^n [y_i - f(x_i)]^2} \quad (3-20)$$

For most of the calibrations, the calibration function was linear ($y = mx + c$). Therefore, the value of the degree of freedom v is equal to two.

For a measured quantity associated with other measured quantities (such as mass flow rate), overall uncertainty u_Y of a parameter $Y = f(x_1, x_2, \dots, x_n)$ is calculated using the following equation:

$$u_Y = \sqrt{\left(\frac{\partial Y}{\partial x_1}\right)^2 u_{x_1}^2 + \left(\frac{\partial Y}{\partial x_2}\right)^2 u_{x_2}^2 + \dots + \left(\frac{\partial Y}{\partial x_n}\right)^2 u_{x_n}^2} \quad (3-21)$$

The instrumentation and overall uncertainties of the measured quantities are summarized and presented in Table 3.10 below.

Table 3.10: Uncertainty of measurements

Physical Location	Measurement	Instrumentation	Uncertainty
V-cone flow meter	Upstream Pressure	Druck PDCR 23D	$\pm 257\text{Pa}$
	Pressure difference	Druck PDCR 22	$\pm 120\text{Pa}$
	Upstream Temperature	E-type thermocouple	$\pm 0.67\text{K}$
	Mass flow	v-cone flow meter	$\pm 1.62\text{g/s}$
Measurement plane (Turbine inlet)	Pressure	Druck PDCR 23D	$\pm 238\text{Pa}$
	Temperature	T-type thermocouple	$\pm 0.5\text{K}$
Turbine exit	Pressure	Druck PDCR 23D	$\pm 238\text{Pa}$
Dynamometer	Rotational Speed	Omron EE-SX1103	$\pm 7\text{rpm}$
	Torque	Interface Miniature Beam	$\pm 0.062\text{Nm}$
Nozzle flow meter (for injection flow)	Pressure difference	Druck PDCR 22	$\pm 120\text{Pa}$
	Upstream pressure	Druck PDCR 23D	$\pm 238\text{Pa}$
	Mass flow	ISO-5167 long radius nozzle	$\pm 2.08\text{g/s}$
	Downstream Temperature	K-type thermocouple	$\pm 1.0\text{K}$
Secondary inlet (injection)	Pressure	Druck PDCR 23D	$\pm 238\text{Pa}$

3.7 Performance Calculations

To assess the performance of the turbine with injection, calculations for the turbine parameters have to be made based on the measured data. The static pressure and temperature can be directly measured, but other parameters, such as the total pressure and total temperature, have to be calculated using the mass flow measurements.

The parameters used for injection are also different from the standard performance parameters used, due to the addition of the injected flow. The following sections will deal with the performance parameters used in the experiments and CFD for the current study.

3.7.1 Mass Flow Parameter

The mass flow parameter of a conventional turbine is defined as the ratio between the actual mass flow rate to a reference mass flow through the turbine at a velocity equal to the stagnation speed of sound, and is given by the expression $\frac{\dot{m}\sqrt{T_{03}}}{P_{03}}$ [14], where station 3 refers to the turbine inlet.

For injection, the reference flow will be a flow with the injection total temperature and pressure passing through the same turbine at the stagnation speed of sound. Therefore, the MFP for a turbine with injection can be expressed as:

$$MFP_{total} = \frac{\dot{m}_{main}\sqrt{T_{03}}}{P_{03}} + \frac{\dot{m}_{inj}\sqrt{T_{0,inj}}}{P_{0,inj}} \quad (3-22)$$

However, from an engine point of view, if the source of injection is not from the engine exhaust gas, then the additional mass flow of the injection is not ‘seen’ by the engine, and in this case the conventional way of expressing MFP should be used. Using the above expression for MFP might give a wrong impression that the engine air mass flow is increased when it is not. Thus, if the MFP is considered for engine performance, some discretion should be exercised as to which expression is more appropriate.

3.7.2 Pressure Ratio

The pressure ratio of the turbine is defined as the ratio between the total pressure at the turbine rotor inlet and the static pressure at the turbine rotor outlet, where stations 3 and 4 refers to the turbine inlet and outlet respectively:

$$PR = \frac{P_{03}}{P_4} \quad (3-23)$$

3.7.3 Isentropic Velocity Ratio

The isentropic velocity ratio is traditionally defined as the ratio between the rotor tip velocity to the velocity achieved through an isentropic expansion through the turbine at the given pressure ratio [14]. The VR is usually used instead of the PR to plot against efficiency as it is less sensitive to changes in Mach number [14]. The isentropic velocity ratio is given by:

$$VR = \frac{U_3}{C_{is}} = \frac{U_3}{\sqrt{2\Delta h_s}} \quad (3-24)$$

Where

$$\Delta h_{is} = \frac{\dot{W}_{is}}{\dot{m}_{total}} \quad (3-25)$$

Here, U_3 is the turbine inlet blade tip speed and $\dot{W}_{is}$ is the isentropic power of the turbine. Therefore, for a turbine with secondary flow injection, the isentropic velocity ratio is given by:

$$VR = \frac{U_3}{\sqrt{2 \frac{\dot{m}_{main} C_p (T_{03} - T_{4s}) + \dot{m}_{inj} C_p (T_{0,inj} - T_{4s,inj})}{\dot{m}_{total}}}} \quad (3-26)$$

where C_p is the specific heat capacity of the fluid at constant pressure.

Thus, the isentropic total to static enthalpy drop here is a mass weighted average of the specific isentropic enthalpy drop between the main flow and the injected flow.

3.7.4 Isentropic Efficiency

The isentropic efficiency of a turbine is defined as the ratio of the actual work of the turbine to the isentropic work. Usually for turbines a total-to-static efficiency is used, since the exit kinetic energy is lost and not recovered. For a turbine with injection, the isentropic work of the injected fluid needs to be taken into account as well. Thus, the

total-to-static isentropic efficiency of a turbine with secondary flow injection is given as:

$$\eta_{TS} = \frac{\dot{W}_{act}}{\dot{m}_{main}C_p(T_{03} - T_{4s}) + \dot{m}_{inj}C_p(T_{0,inj} - T_{4s,inj})} \quad (3-27)$$

where given a certain angular speed ω ,

$$\dot{W}_{act} = Torque \times \omega \quad (3-28)$$

3.7.5 Entropy Generation (CFD)

The entropy generation rate is a useful tool to locate sources of losses in turbomachinery. The entropy generation rate can be computed during post-processing in CFD, using the direct calculation method of Herwig and Kock [15], which will be briefly explained in following.

The total entropy generation rate per unit volume is comprised of four components:

$$\dot{s}_{gen} = \dot{s}_{gen,\bar{D}} + \dot{s}_{gen,D'} + \dot{s}_{gen,\bar{C}} + \dot{s}_{gen,C'} \quad (3-29)$$

where $\dot{s}_{gen,\bar{D}}$ is the entropy generation by viscous dissipation of the mean flow field, $\dot{s}_{gen,D'}$ is the entropy generation by turbulent dissipation of the flow, $\dot{s}_{gen,\bar{C}}$ is the entropy generation by heat conduction with mean temperature gradients, and $\dot{s}_{gen,C'}$ is the entropy generation by heat conduction with fluctuating temperature gradients.

$\dot{s}_{gen,\bar{D}}$, the entropy generation due to viscous dissipation of the mean flow field, can be directly determined:

$$\begin{aligned} \dot{s}_{gen,\bar{D}} = \frac{\mu}{\bar{T}} \left\{ 2 \left[\left(\frac{\partial \bar{u}}{\partial x} \right)^2 + \left(\frac{\partial \bar{v}}{\partial y} \right)^2 + \left(\frac{\partial \bar{w}}{\partial z} \right)^2 \right] + \left(\frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x} \right)^2 \right. \\ \left. + \left(\frac{\partial \bar{u}}{\partial z} + \frac{\partial \bar{w}}{\partial x} \right)^2 + \left(\frac{\partial \bar{v}}{\partial z} + \frac{\partial \bar{w}}{\partial y} \right)^2 \right\} \end{aligned} \quad (3-30)$$

where μ is the dynamic viscosity of the fluid.

Similarly, $\dot{s}_{gen,\bar{C}}$, the entropy generation due to heat conduction with mean temperature gradients, can be directly determined:

$$\dot{s}_{gen,\bar{C}} = \frac{\lambda}{\bar{T}^2} \left[\left(\frac{\partial \bar{T}}{\partial x} \right)^2 + \left(\frac{\partial \bar{T}}{\partial y} \right)^2 + \left(\frac{\partial \bar{T}}{\partial z} \right)^2 \right] \quad (3-31)$$

where λ is the thermal conductivity of the fluid.

However, the two turbulent terms $\dot{s}_{gen,D}$ and $\dot{s}_{gen,C}$ cannot be directly determined, and requires turbulence modelling. Herwig and Kock proposed the following turbulence models for $\dot{s}_{gen,D}$ and $\dot{s}_{gen,C}$:

$$\dot{s}_{gen,D} = \frac{\rho \varepsilon}{\bar{T}} \quad (3-32)$$

$$\dot{s}_{gen,C} = \frac{\alpha_t}{\alpha} \dot{s}_{gen,\bar{C}} \quad (3-33)$$

where ρ is the density, ε is the turbulent dissipation rate, α_t is the turbulent thermal diffusivity, and α is the thermal diffusivity.

All the above terms can be obtained in post-processing of the CFD results, and thus the overall entropy generation rate can be computed. In order to compare results between different cases where the mass flow rates and thus the overall entropy generation might differ (e.g. baseline and injection), the overall entropy generation rate per unit volume is normalized using the isentropic power of the turbine:

$$\dot{s}_{gen,norm} = \frac{\dot{s}_{gen}}{W_{isen}} \quad (3-34)$$

It is to be noted, however, that while the entropy generation rate can provide very useful information on the regions with high losses, the two-equation CFD models currently employed in this thesis will not accurately quantify the entropy generation [3]. Therefore, the entropy generation rate will only be used to make qualitative analysis in this thesis.

References

- [1] M. Abidat, "Design And Testing Of A Highly Loaded Mixed Flow Turbine," Doctor of Philosophy, Imperial College London, 1991.
- [2] S. Rajoo, "Steady and pulsating performance of a variable geometry mixed flow turbocharger turbine," Doctor of Philosophy, Imperial College London, 2007.
- [3] P. Newton, T. Palenschat, R. Martinez-Botas, and M. Seiler, "Entropy generation rate in a mixed flow turbine passage," in *International Gas Turbine Congress 2015*, Tokyo, Japan, 2015.
- [4] ANSYS, "ANSYS Help," ed, 2017.
- [5] MathWorks. (2019). *Genetic Algorithm - MATLAB & Simulink*. Available: <https://www.mathworks.com/discovery/genetic-algorithm.html>
- [6] U. Khairuddin, A. W. Costall, and R. F. Martinez-Botas, "Influence of geometrical parameters on aerodynamic optimization of a mixed-flow turbocharger turbine," presented at the ASME Turbo Expo 2015: Turbine Technical Conference and Exposition, 2015.
- [7] D. Japikse and N. Baines, *Introduction to turbomachinery*. 1994.
- [8] W. Bin Wan Salim, "Study of externally waste-gated turbine performance under steady and pulsating inlet conditions for improved turbocharger matching," Doctor of Philosophy (PhD), Mechanical Engineering, Imperial College London, 2014.
- [9] McCrometer, "V-Cone Installation Operation & Maintenance Manual," ed, 2017.
- [10] BS EN ISO 5167, "Measurement of fluid flow by means of pressure differential devices inserted in circular cross-section conduits running full," ed, 2019.
- [11] J. F. C. Forés, "Effect of inlet configuration and pulsation on turbocharger performance for enhanced energy recovery," PhD, Department of Mechanical Engineering, Imperial College London, 2018.
- [12] L. Kirkup, *Experimental Methods for Science and Engineering Students: An Introduction to the Analysis and Presentation of Data*, Second ed. Cambridge University Press, 2019.
- [13] P. J. Newton, "An Experimental and Computational Study of Pulsating Flow within a Double Entry Turbine with Different Nozzle Settings," PhD, Department of Mechanical Engineering, Imperial College London, 2014.
- [14] A. Whitfield and N. C. Baines, *Design of Radial Turbomachines*. Longman Scientific & Technical, 1990.
- [15] H. Herwig and F. Kock., "Direct and indirect methods of calculating entropy generation rates in turbulent convective heat transfer problems," *Heat and mass transfer* vol. 43, no. 3, pp. 207-215, 2007.

Chapter 4: Single Passage Simulations and Optimization

This chapter presents the results of the single passage CFD simulations, as well as optimization results for the injection parameters. The baseline CFD simulation results show that the turbine efficiency peaks at 76% at an isentropic velocity ratio (VR) of 0.63. Flow visualization shows the presence of a large separation vortex on the suction side at a range of operating points. At high VR, separation on the pressure side also occurs. Initial injection simulations that focused on flow near the blade tip showed increased turbine power output, but efficiency improvements were only obtained at high VR. The injected flow improved blade tip loading by strengthening the tip leakage vortex, but efficiency gains were minimal. The initial simulations also found that injection velocity and yaw angle were important parameters that affect turbine performance. Optimization of the injection parameters was then performed at peak efficiency point, 30,000rpm. The optimized injection flow was directed further into the turbine passage, and results showed an efficiency increase of 2.6 percentage points at the optimized point. The same injection parameters also resulted in efficiency improvements across the entire speed line. Flow visualization shows that the injected flow forces the passage flow to migrate towards the hub, creating a partial blockage and forces the passage flow to more closely follow the streamwise direction. This weakens the suction side hub separation vortex and thus reduces the losses induced.

*This section published/submitted substantially as [1-3].

4.1 Baseline Simulations

4.1.1 Turbine Performance Curves

Figure 4.1 shows the performance curves of the baseline turbine obtained from the single passage CFD simulations. The operating points 1, 5, and 8 (according to Table 3.4) are also labelled for convenience. The results show typical performance characteristics of a mixed flow turbine. The pressure ratio ranges between 1.1 to 1.5 for the points simulated, while the mass flow parameter ranges between $2.1e^{-5}$ and $4.8e^{-5}$. The isentropic velocity ratio ranges from 0.49 to 0.98. The turbine efficiency peaks at 76% at an isentropic velocity ratio of 0.63. This is expected of a mixed flow turbine, where the peak efficiency point should occur at an isentropic velocity ratio of less than 0.7.

To better understand the aerodynamics of the mixed flow turbine, detailed flow visualization results are presented in the following section.

4.1.2 Detailed Flow Analysis

Figure 4.2 and Figure 4.3 show the flow within the rotor passage for the baseline cases for points 1, 5 and 8 on the suction surface and pressure surface respectively. The streamlines are coloured by the normalised entropy generation rate.

Figure 4.1: Performance Curves of Baseline Turbine, Single Passage CFD

Figure 4.2: Streamlines within passage for baseline, suction side. Top: Point 1, Middle: Point 5, Bottom: Point 8

The figures show that the flow structures are rather different in Point 1 as compared to the other 2 points. At Point 1, there is a large separation vortex on the suction surface of the blade at about 2/3 chord length and 25% span, labelled A. A separation vortex appears on the suction surface in points 5 and 8 labelled B, but they are closer to the

leading edge and on the suction-surface-hub corner. This difference can be attributed to the difference in incidence between point 1 and the other 2 points. From Figure 4.2, it can be seen that there is negative incidence on the leading edge of the blade at point 1, but positive incidence at points 5 and 8, as shown by the velocity triangles. Therefore, for point 1, no separation occurs on the suction surface near the front of the blade, as the incidence angle is favourable to the suction surface. As the flow approaches nearer the aft of the blade, it approaches an unfavourable pressure gradient and separates, and this separated flow is turned towards the shroud and away from the suction surface due to the centrifugal and Coriolis forces, developing into the separation vortex A. For points 5 and 8, the positive incidence angle causes separation to occur very near the leading edge, and the resulting separation vortex does not migrate as much away from the suction surface, presumably because the higher meridional velocity of the fluids help keep the flow stay in direction. Also to be noted is that a smaller vortex B is formed right above the hub separation vortex in points 5 and 8, which soon merges with the hub separation vortex. For point 8, this actually consists of 2 vortices B1 and B2, shown by the surface streamlines in Figure 4.4. The mechanism for the formation of this smaller vortex structure is not yet clear, but its apparent contribution to overall entropy generation is not significant. There is also a clear hub to shroud secondary flow nearer to the trailing edge. This is similar to the results found in [4].

Figure 4.3: Streamlines within passage for baseline, pressure side, Point 1

The negative incidence in point 1 also results in separated flow on the pressure surface, which then reattaches and migrates towards the shroud and then rolls up into another separation vortex. This is labelled C1 and C2 on Figure 4.3. There are 2 main features

of this vortex. Feature C1 is the fluid originating from the hub that is entrained by the vortex and migrates to the shroud. This fluid then turns due to the vortex and hits the suction surface of the adjacent blade. This is labelled C on Figure 4.2. This stream of fluid then migrates towards the hub and rolls up into the suction side separation vortex. Feature C2 in Figure 4.3 is the core of the pressure side separation vortex. At points 5 and 8, as the incidence angle is positive, no such separation occurs on the pressure surface, and therefore the streamlines are not shown.

Figure 4.4: Surface streamlines for baseline, suction side, Point 8

The normalized entropy generation contours are shown in Figure 4.5 for the suction side. The figure shows a few main areas with high entropy generation. For point 1, the high entropy generation area associated with the suction surface separation vortex can be clearly seen, labelled A. In addition, there is a region of high entropy generation associated with the pressure side separation vortex, labelled C1 and C2. Region C1 corresponds to the strong secondary flow migrating from the adjacent pressure surface shown as C in Figure 4.2. Comparing region C2 with the streamlines in Figure 4.2 seem to indicate that this is where the suction side separation vortex mixes with the core of the pressure side separation vortex which has also migrated towards the adjacent suction surface. The third region of high entropy generation rate is the tip leakage vortex, labelled D. The tip leakage vortex is relatively small in size at point 1. For point 5 and point 8, the regions of high entropy generation rate associated with the suction surface separation vortex and the tip leakage flow can also be seen, labelled A and D respectively. For points 5 and 8, one can see the high entropy region associated with the tip leakage vortex moving upstream. The smaller vortex structures above the suction

side separation vortex in points 5 and 8 (feature B in Figure 4.2) do not seem to contribute to significant entropy generation.

Figure 4.5: Contours of entropy generation rate for baseline, suction side. Top: Point 1, Middle: Point 5, Bottom: Point 8

On the pressure side, there are 2 main regions of high entropy generation at point 1. Both of these are associated with the pressure side separation vortex. These are shown as regions C1 and C2 in Figure 4.6. Region C1 corresponds to the secondary flow from hub to shroud that turns towards the adjacent suction surface (feature C1 in Figure 4.3), while region C2 is associated with the core of the pressure side vortex. From the contours, it can be seen that the losses associated with the pressure side separation vortex is very significant.

Figure 4.6: Contours of entropy generation rate for baseline, pressure side, Point 1

4.2 Initial Injection Simulations

For injection, as a starting point of investigation, some initial values for the injection parameters had to be decided to explore the effectiveness of the concept. The injection slot was chosen to be closer to the trailing edge of the blade because tip leakage flow is most prominent in this section, and the width of the slot was chosen to provide a reasonable injection velocity that is higher than the passage flow velocity but still subsonic. An injection mass flow rate of 5% passage mass flow was imposed. The flow was injected in such a way that it remains attached to the shroud surface so as not to mix with or block the primary passage flow. The injected flow has a total temperature of 333K, similar to the inlet total temperature of the main passage flow. The injection pressure was found to range between 104.5kPa to 110.0kPa, depending on the injected mass flow. A pitch angle of 15 degree was imposed so as to ensure that the secondary flow is injected at a slight angle to the slot surface to prevent excessively high jet velocities from having an injection almost parallel to the injection slot. The flow was

also designed to stay close to the injection slot so that its effects on the tip leakage flow can be assessed. The effects of injection yaw angle are investigated. The effect of secondary injection on turbine performance is assessed through the effects of injection on both turbine work and isentropic efficiency. A total of 4 different injection yaw angles θ were tested: 0, 30, 45 and 60 degrees. For each of the non-zero yaw angles an additional simulation was also conducted whereby the injection mass flow is reduced to keep the same injection velocity as the 0° yaw case (since the area of the slot is constant, at the same injection mass flow, any change in injection direction will increase the injection flow velocity because the normal component of the injected flow must be kept constant). The reduced mass flow is given by:

$$\dot{m}_{reduced} = \cos\theta * \dot{m}_{inj,\theta=0} \quad (4-1)$$

This is an attempt to separate the effects of the jet velocity from the jet direction. . All the simulated case studies are reported in Table 4.1.

Table 4.1: Summary of Injection Parameters Tested, Preliminary Investigation

Mass flow rate	5% passage mass flow
Jet Velocity [m/s]	70 (lowest mass flow, 0 yaw) – 332 (highest mass flow, 60 yaw)
Injection total temp [K]	333
Injection pressure [kPa]	104.5-110.0
Slot location from LE tip [mm]	14.0
Slot size [mm]	2.0
Pitch	-15°
Yaw	0°, 30°, 45°, 60°

4.2.1 Varying Yaw Angle

The effect of varying yaw angle on turbine performance is studied in the following section. The effect of the injected secondary flow on the blade loading profiles, as well as other flow structures, are presented.

Turbine Power

Figure 4.7 shows the change in turbine power over the baseline case due to secondary injection. From the figure, it can be seen that generally, there is an increase in turbine power from secondary injection, with more increases at higher velocity ratios. It can

also be seen that as the injection yaw angle increases, the increase in turbine power is larger, with 60° yaw injection resulting in more than 10% increase in power at velocity ratios larger than 0.8. The results suggest that injection with a positive yaw angle (in the same direction as turbine rotation and tip leakage flow) actually improves the turbine blade loading, which seems to be counter-intuitive, since this seems to promote tip leakage flow.

Figure 4.7: Change in turbine power due to injection over baseline scenario

Figure 4.8 shows the blade loading diagrams for injection at different yaw angles compared to baseline at 95% span, for an isentropic velocity ratio of 0.7. From Figure 4.8, it can be seen that the secondary injection has two main effects on the blade tip loading:

1) The first effect is on the blade suction surface. Clearly, there is a drop in pressure on the suction surface from blade streamwise direction 0.5 onwards with secondary injection. As the yaw angle increases, the pressure drop is higher, but the pressure recovers earlier along the streamwise direction as well. The drop in pressure can be attributed to a strengthening of the leakage vortex, as seen in Figure 4.9. The leakage vortex has a low pressure core, as shown in the lower pressure zones that coincide with the leakage vortex streamlines (purple). The secondary injection intensifies the vortex, thus decreasing the pressure of the vortex core. Since the leakage vortex is close to the

suction surface, this improves the pressure loading on the suction surface. This phenomenon demonstrates clearly the potential in conditioning the leakage vortex to improve turbine performance, which is a novel idea not found in any present literature. However, it can also be seen that as the yaw angle increases, the leakage vortex is also deflected away from the suction surface nearer to the trailing edge, which explains earlier pressure recovery on the suction surface.

2) The second effect of injection is that due to the injected flow having a component that is being directed towards the blade pressure surface, the injected flow entrains the passage flow in its vicinity to ‘push’ the turbine blade directly. This results in higher pressure on the blade pressure surface from blade streamwise direction 0.6 onwards.

The combination of the abovementioned mechanisms results in a lower pressure on the suction surface and higher pressure on the pressure surface, improving the overall blade loading.

Figure 4.8: Blade loading at 95% span for injection at different yaw angles compared with baseline, $VR=0.7$

Figure 4.9: Tip leakage vortex pressure contours and streamlines at $VR= 0.7$ for: a)baseline, b)injection 0° yaw, c)injection 30° yaw, d)injection 45° yaw

Turbine Efficiency

Figure 4.10 shows the change in turbine efficiency due to the secondary injection. From the figure, it can be seen that the significant improvements in turbine power does not fully translate into higher turbine efficiencies. In fact, at velocity ratios below 0.8, there is negative efficiency gains for all cases. The maximum efficiency increase is 1.9 percentage points, which was achieved at 45° yaw.

The low net efficiency gains as compared to the high power gains is mainly due to the fact that the injection has a high energy cost which was factored into the efficiency calculations. The secondary injection system generates additional losses through viscous shearing due to the higher velocity as compared to the main passage flow. Nonetheless, an efficiency improvement of 1.9 percentage points at higher velocity ratios is significant, especially considering that the turbine operates at 60% or lower efficiency in these conditions. The abrupt increase in turbine efficiency from $VR = 0.76$ to $VR = 0.86$ is partly attributed to the greater increase in turbine power.

Figure 4.10: Change in turbine efficiency due to injection over baseline scenario

4.2.2 Varying Yaw Angle with Reduced Mass Flow

When the yaw angle is increased, since the mass flow and the actual area in which the injected jet flows through is the same, the effective area that the injected jet experiences is reduced. As a result, the jet velocity increases. To assess the effect of yaw angle while keeping the jet velocity constant, the injection mass flow is reduced accordingly. The following section discusses the results of injection whereby the injected mass flow is reduced to maintain the same jet velocity at different yaw angles.

Turbine Power

Figure 4.11 shows the change in turbine power for secondary injection at different yaw angles with mass flow adjusted to maintain the same injection velocity. It can be seen that the overall increase in turbine power is fairly similar for all 4 cases. For injection at 60° yaw, the increase in turbine power is even lower than injection at lower yaw angles at higher velocity ratios. This suggests that the previous larger improvements in turbine power from larger yaw angles are more likely due to higher energy being imparted from a jet with higher mass flow and velocity rather than a direct result of the yaw angle itself.

The blade loading chart in Figure 4.12 shows that when the injection velocity is maintained constant, injection at different yaw angles does not have significant effect on the blade tip loading. This is also seen in the pressure contours in Figure 4.13. The

strength of the leakage vortex does not seem to have changed significantly for the 3 cases with secondary injection.

The relative insensitivity of the blade loading to injection yaw angle when injection mass flow is reduced to maintain the same injection velocity seems to suggest that injection velocity is a much more important factor than injection yaw angle in influencing the flow conditions near the blade tip.

Figure 4.11: Change in turbine power compared to baseline scenario for reduced mass flow case

Turbine Efficiency

Figure 4.14 shows the efficiency changes compared to baseline when secondary flow is injected at different yaw angles while the mass flow is reduced to maintain the same injection velocity. From the figure, it can be seen that all 3 cases with positive yaw angle exhibit higher efficiencies than injection without yaw. Also, there is a clear improvement in efficiency at velocity ratios less than 0.8 as the yaw angle increases, while at velocity ratios more than 0.8, injection at 45° yaw has slightly better efficiency than other injection yaw angles.

Figure 4.12: Blade loading at 95% span for injection with reduced mass flow at different yaw angles, compared with baseline, $VR=0.7$

Figure 4.13: Tip leakage vortex pressure contours and streamlines at $VR= 0.7$ for: a)baseline, b)injection 0° yaw, c)injection 30° yaw (mass flow reduced), d)injection 45° yaw (mass flow reduced)

The higher efficiencies at higher yaw angles can be attributed to the fact that an increase in injection yaw angle is accompanied by a decrease in injection mass flow rate. Thus, the injection kinetic energy is reduced as injection yaw angle increases, if the injection velocity is maintained constant. This reduces the isentropic work done by the system, and since the actual turbine work does not change much as mentioned previously, the efficiency increases.

Figure 4.14: Change in efficiency compared to baseline scenario for reduced mass flow case

The results show that injection is able to improve turbine efficiency at higher velocity ratios. To improve the results at other operating points, optimization was conducted.

4.3 Optimization Results

Table 4.2 below shows the final optimized parameters. The optimized point has an injection mass flow of 0.0395kg/s, at an injection stagnation pressure of 117kPa. This results in an increase in efficiency increase of 2.6 percentage points over the baseline scenario at the simulated operating point. It is to note that three of the injection parameters are at their lower or upper bound values, which implies that there are likely other points outside the current design space which might provide even higher

efficiencies than the current value. This will be interesting to explore in future work, as the current computational mesh is likely to fail if the location and width values are changed beyond the current bound values. Also, it can be seen that the injection yaw angle is already 80° . This already requires the injected flow to turn to a very large circumferential angle. It will be physically difficult to inject fluid at even higher circumferential angles. The optimized pitch angle is 60° , which means that the injected flow is expected to interact with the turbine passage flow not only near the blade tip, but further down the span as well.

Table 4.2: Final Optimized Parameters

Parameter	Final Optimized	Baseline
Location (mm)	5.5	-
Width (mm)	5.0	-
Velocity (m/s)	116	-
Yaw ($^\circ$)	80	-
Pitch ($^\circ$)	60	-
Efficiency	0.78644	0.76035

To assess the impact of the optimized injection flow over a range of operating conditions, the optimized injection parameters were used to create a speed line using the operating points in Table 3.4. Figure 4.15 shows the efficiency against VR curves for the optimized injection compared with baseline. The graph shows that in general, injection increases the total to static isentropic efficiency throughout all the operating points, with a 2.6 percentage point efficiency increase at the optimised peak efficiency point (Point 5). The increase in efficiency is larger at higher VR (lower mass flow rate).

The effect of injection on the PR vs MFP curve can be seen in Figure 4.16. As the additional mass flow injected into the turbine at a location partway through the turbine expansion process, the injected mass flow does not result in significant increase in the turbine inlet pressure. As a result, the PR vs MFP curve is shifted to the right i.e. there is an increase in the turbine swallowing capacity. If the injection source is external, then the engine would ‘see’ a slight blockage in the MFP characteristic, shown as the dotted

blue line labelled ‘altMFP’ in Figure 4.16, but even so the blockage effect is only marginal.

Figure 4.15: Turbine efficiency curve for baseline and injection cases

Figure 4.16: Turbine Pressure Ratio VS MFP Curve for Baseline and Injection Cases

4.3.1 Detailed Flow Analysis

Detailed flow visualization was conducted to assess how the optimized injection affects the flow field at the optimized point, as well as other points. Figure 4.17 shows the flow

structures of the rotor passage with injection. For point 1, the effects of injection on the suction surface separation vortex is very clear. The injected fluid enters from the blade tip towards the hub, inhibiting the formation of the separation vortex, as seen labelled A'. However, the pressure side separation vortex seems to have been given momentum by the injection, as seen in label C2', where the streamlines of the vortex are more structured and close together. The pressure side separation vortex and the strong secondary flow migration associated with it (C2' and C1' respectively) were also moved towards the hub. Injection has also been seen to weaken the suction side separation vortex in points 5 and 8 as well, though it can be seen to be less effective as the turbine mass flow increases. This is within expectations, since a higher passage mass flow means higher velocities which have higher momentum, and the same amount of injection will have less effect.

The injected fluid also creates a partial blockage in the rotor passage, evident in Figure 4.17 in the regions labelled E, where the streamlines have been deflected towards the hub to form a pocket. This forces the flow through the passage through a smaller area, and thus forces the passage flow to more closely follow the streamwise direction. This likely helps in weakening the suction side separation vortex at all 3 operating points.

The effects on the flow features on the pressure side for point 1 are already mentioned, while there is no significant change in flow on the pressure side for points 5 and 8 due to injection, so the streamlines on the pressure side will not be presented.

Looking at the entropy generation contours, shown in Figure 4.19, the effects of injection on the rotor losses can be seen. For point 1, a significant change is the reduction in entropy generation rate due to the suction side separation vortex, labelled A' in Figure 4.19. Compared with the baseline in Figure 4.5, the losses associated with this vortex has almost completely been erased. However, there is still significant entropy generation due to the pressure side separation vortex, as can be seen in label C', although injection has reduced the entropy generation rate associated with the pressure side separation vortex. This is evident in the lack of high entropy generation found in regions C1 and C2 in Figure 4.5. Looking at the blade tip, injection seems to have increased the high-loss area related to tip leakage flow, evident by the larger area of D'. On the pressure surface, it is also evident that the high entropy generation regions

associated with the pressure side separation vortex has been reduced and moved towards the hub by the injected fluid. This is shown as regions C1' and C2' in Figure 4.18.

Figure 4.17: Streamlines within passage for injection, suction side. Top: Point 1, Middle: Point 5, Bottom: Point 8

For point 5, the entropy generation contours show that with injection, the high entropy region D' is compressed towards the blade suction surface and elongated towards the hub. Also, the entropy generation due to the suction side separation vortex is reduced slightly as indicated by A', showing that the injection is effective in reducing losses arising from the separation vortex. In addition, closer to the trailing edge of the blade, an additional region of high entropy generation was formed, shown as region E'. This region coincides with the separation of the leakage flow streamlines to form two counter-rotating vortex pairs, shown in Figure 4.20. This shows that near the blade tip, injection is detrimental to turbine efficiency, even though turbine work might be improved through better loading.

Figure 4.18: Contours of entropy generation rate for injection, pressure side, Point 1

For point 8, injection also reduces the entropy generation from the suction surface separation vortex slightly, as indicated by A'. Near the shroud, the high entropy generation region associated with the tip leakage flow D' has been reduced near the trailing edge. Compared to region D in Figure 4.5, where there is a thin region of high entropy generation near the shroud across the entire blade passage close to the trailing edge, region D' in Figure 4.19 shows a significantly reduced high entropy generation area which is confined near the suction surface. This shows that injection is effective in reducing the losses from tip leakage, though the authors are unclear of the exact mechanism on how this happens at this point of time. At point 8, injection also causes the tip leakage flow to separate into a vortex pair, which causes additional losses as indicated by region E'. However, this effect is relatively weak compared to the one in

point 5. Overall, the results show that secondary flow injection is an effective method in improving turbine performance.

Figure 4.19: Contours of entropy generation rate for injection, suction side. Top: Point 1, Middle: Point 5, Bottom: Point 8

Figure 4.20: Tip leakage Flow Structure, Point 5. Left: Baseline, Right: Injection

References

- [1] H. Liu, A. Romagnoli, R. Martinez-Botas, S. Rajoo, and M. H. Padzillah, "Multi-Injection Turbine Housing: A Novel Concept for Tip-Leakage Improvement in Radial Turbines," in *ASME Turbo Expo 2017: Turbomachinery Technical Conference and Exposition*, Charlotte, NC, USA, 2017, pp. V02CT44A022-V02CT44A022: American Society of Mechanical Engineers.
- [2] H. Liu, A. Romagnoli, R. Martinez-Botas, S. Rajoo, and M. H. Padzillah, "Effects Of Secondary Flow Tip Injection On Performance And Losses In A Mixed Flow Turbine," in *2018 Global Power and Propulsion Forum*, Montreal, 2018: Global Power and Propulsion Society, 2018.
- [3] H. Liu, A. Romagnoli, M. I. Ismail, R. Martinez-Botas, S. Rajoo, and M. H. Padzillah, "Multi-Injection Turbine Housing: A Novel Concept For Performance Improvement In Radial Turbines," in *The 13th International Conference on Turbochargers and Turbocharging*, London, 2018: Institution of Mechanical Engineers.
- [4] L. Chen, W. Zhuge, Y. Zhang, S. Zhang, and J. Zhang, "Investigation of the secondary flow structure in the mixed flow turbine for a high pressure ratio turbocharger," presented at the ASME Turbo Expo 2008: Power for Land, Sea, and Air, 2008.

Chapter 5: Full Stage Simulations and Validation With Experimental Results

This chapter presents the results of the full stage CFD validation with experimental results. Results for additional full stage CFD simulations which assess the potential improvements of the turbine through secondary flow injection are also presented. Validation results show that CFD predicts the torque well, while pressure ratio is under-predicted at both 30,000rpm and 48,000rpm. Efficiency was well predicted at low VR for 30,000rpm, but under-predicted at 48,000rpm. Overall, CFD captures the performance trends of the turbine reasonably well, but uncertainty in 48,000rpm results lead to further CFD simulations being done at a lower turbine speed. The full stage injection simulations show that with inlet nozzle vanes, injection did not result in any visible improvement in the internal flow field and entropy generation, but overall efficiency can be improved by up to 2.28% points at a velocity ratio of 0.75. In a vaneless configuration, injection resulted in an efficiency improvement of up to 6.79% points by weakening the hub suction side separation vortex and reducing its associated losses. Injection on the vaneless turbine configuration also resulted in a roughly 2% point improvement in peak turbine efficiency over the vaned configuration. This might be due to more flow energy available from reduced losses in the vaneless turbine.

5.1 Full Stage CFD Validation with Experiments

Figure 5.1 shows the results of the full stage CFD simulations compared with experiments at 30,000rpm. The simulation results are from case A in Figure 3.14. It can be seen that the CFD slightly under-predicts the pressure ratio (PR) at a certain mass flow parameter (MFP) compared to experiment. However, the general PR vs MFP characteristic is captured reasonably well. The torque vs VR curve is predicted very well by the CFD, with a slight over-prediction of torque at low VR.

The isentropic efficiency is predicted well by the CFD at velocity ratios under 0.8. At higher velocity ratios, the CFD under predicts the isentropic efficiency compared to the experiment. This can be seen in the baseline curves. The reason for the discrepancy is due to the slight under-prediction of turbine torque in the CFD simulations, which at such low turbine power will make a significant difference in the efficiency calculations.

For 48,000rpm, the full stage CFD and experimental validation results are shown in Figure 5.2. The PR is still under-predicted at a given MFP, with the under-prediction being more severe for the injection case. This might suggest that the injection introduces higher blockage to the turbine in the experiment which was not captured by the CFD. The overall PR vs MFP trend was, however well captured. The torque vs isentropic velocity ratio (VR) curve is predicted very well by the CFD, though there is a slight under-prediction of turbine torque at high velocity ratios. The isentropic efficiency is generally under-predicted by CFD. Also, in the CFD results there was no peak efficiency, while in the experiments the peak efficiency occurs at a VR of around 0.66 for baseline and 0.64 for injection. This might be a problem of the flow solver, as even after running simulations at lower VR (higher PR), the efficiency continued to increase (results not shown). The turbine inlet flow at $VR < 0.65$ is significantly transonic, and this might cause some problems for the compressible flow solver used in CFX, and gives rise to erroneous torque outputs which affect the efficiency results. This uncertainty in the CFD results at 48,000rpm lead to the following investigations all being done at a lower turbine speed.

Figure 5.1: 30,000rpm CFD validation results: Top) Pressure Ratio (PR) against Mass Flow Parameter (MFP) Middle) Torque against Velocity Ratio (VR) Bottom) Efficiency against Velocity Ratio (VR)

Figure 5.2: 48,000rpm CFD validation results: Top) Pressure Ratio (PR) against Mass Flow Parameter (MFP) Middle) Torque against Velocity Ratio (VR) Bottom) Efficiency against Velocity Ratio (VR)

5.2 Effects of Injection in Full Stage Simulations

From the full stages CFD and the experimental results (Figure 5.1), it can be noted that injection results in a lower efficiency compared to the baseline, which contradicts the preliminary results. From the full stage simulations it was found that there were two contributing factors. Firstly, the injection housing used in the experiment and the full stage simulations was an unrefined design and experienced significant aerodynamic losses. Also, the housing was not able to provide the exact injection conditions as the single passage simulations. This was further complicated by the fact that there was an error in the mass flow measurement of the injected flow during experiment, which resulted in an injection mass flow rate which was lower than intended.

Secondly, due to the presence of inlet nozzle vanes in the full turbine, the inlet flow field to the rotor blades was very different from the single passage model. Notably, the suction side separation vortex present in the single passage simulations was largely absent. Thus, the benefits of the injection in reducing losses from this separation vortex was not realized. This can be seen in Figure 5.3, for operating point 5.

Figure 5.3: Rotor Streamline Comparison: Left) Single Passage CFD, Right) Full Stage CFD

5.3 Potential Improvements Through Injection

Additional investigations (see Figure 3.14) were conducted to assess the effectiveness of secondary flow injection in different scenarios. The following section discusses some

of the potential improvements of injection using the results of full stage CFD simulations.

Figure 5.4 shows CFD cases whereby losses from the injection duct were ignored (cases A and C), or if flow is injected as a boundary condition (cases B and D). Efficiency at a few representative velocity ratios were also included in the table for reference. If the losses from the injection housing were neglected in the calculations, it can be shown that turbine efficiency was comparable to that of the baseline at most operating points, for both injection mass flow of 33g/s (case A) and 40g/s (case C), except for velocity ratios less than 0.5. Furthermore, for injection mass flow of 40g/s, although it was the optimized injection mass flow from single passage simulations, its efficiency was lower than both case A and the baseline at velocity ratios above 0.7. This shows that with the injection duct, the flow has deviated enough from the ideal injection conditions that the intended injection mass flow rate no longer results in the best efficiency.

In the case of ‘perfect’ injection conditions (i.e. injection as boundary condition, cases B and D), it can be seen that there is a slight efficiency increase compared to baseline, especially at higher velocity ratios. At a velocity ratio of 0.6, cases B and D showed a 0.95% and 0.93% point efficiency increase over the baseline respectively, and the improvement was 1.97% and 2.28% points respectively at a velocity ratio of 0.75. Cases B and D also show little difference in performance with each other despite the difference in injection mass flow, which suggests that if the direction of injection is perfect, some differences in the injection mass flow can be tolerated without much penalty in performance.

Figure 5.5 and Figure 5.6 provide a visualization of the flow from the baseline full stage CFD simulations, as well as cases C and D at operating point 5 (refer to Table 3.7). Flow streamlines from Figure 5.5 show that with a vaned turbine at point 5 (close to $VR=0.6$), there is little change in the streamlines within the rotor, except for the injected flow forcing flow nearer to the blade tip towards the hub. Entropy generation contours from Figure 5.6 show little difference between the 3 cases as well, so the mechanism which efficiency was improved was not clear. The results show that overall, there might be a small potential for efficiency improvement through flow injection, but the reason is not well understood at this point. However, practically it is difficult to achieve

this improvement, since there are bound to be inefficiencies in the injection system that cancel out the improvements.

Figure 5.4: Efficiency Curves of Different Vaned Turbine Injection Cases Compared with Baseline, 30,000rpm

Figure 5.5: Velocity Streamlines for Vaned Turbine: left) Baseline Vaned Full Stage, Middle) Injection Case C, Right) Injection Case D

Figure 5.6: Normalized Entropy Generation Rate Contours for Vaned Turbine: left) Baseline Vaned Full Stage, Middle) Injection Case C, Right) Injection Case D

Since the previous simulations show that it is difficult to achieve an efficiency improvement on a vaned turbine as in the experiment, it was hypothesized by the authors that injection will be effective in vaneless set ups. The results from the vaneless simulations, cases E and F, are shown in Figure 5.7. From the results, a clear improvement in turbine efficiency could be seen in the injection simulations over the baseline vaneless case. At a lower velocity ratio of 0.5, the simulation ignoring losses from the injection duct (case E) showed minimal improvement in efficiency of 0.42% points, while the simulation for a ‘perfect’ injection (case F) showed a larger 1.51% points improvement. At a velocity ratio of 0.65, which is near peak efficiency for both case E and F, case E showed an efficiency improvement of 3.4% points, while case F showed an efficiency improvement of 4.34% points. At a higher velocity ratio of 0.75, case E showed an efficiency improvement of 3.06% points, while case F showed an efficiency improvement of 6.79% points. The results show that in the current configuration, secondary flow injection can be an effective way in improving vaneless turbine efficiency. The efficiency improvement can be attributed to the injected flow weakening the suction side separation vortex, similar to the single passage simulations shown previously. This can be seen in the regions of interest marked in a box in Figure 5.8 and Figure 5.9 for point 5 ($0.62 < VR < 0.66$ for the 3 cases), in the streamlines and entropy generation rates respectively.

Figure 5.7: Efficiency Curves of Different Vaneless Turbine Injection Cases Compared with Baseline, 30,000rpm

Figure 5.8: Velocity Streamlines for Vaneless Turbine: left) Baseline Vaneless Full Stage, Middle) Injection Case E, Right) Injection Case F

Figure 5.9: Normalized Entropy Generation Rate Contours for Vaneless Turbine: left) Baseline Vaneless Full Stage, Middle) Injection Case E, Right) Injection Case F

However, the current configuration is obviously not realistic, as vaneless turbines usually have much smaller interspaces between the volute exit and rotor inlet. Thus, future study on the effects of secondary flow injection on a more realistic vaneless turbine design should be conducted.

It is also interesting to note that injection results in a roughly 2% point higher turbine peak efficiency in the vaneless configuration than the vaned one. The reason for this higher performance was traced back to a higher turbine torque in the vaneless configuration, which resulted in higher actual turbine work, while isentropic work remained similar. The increase in torque produced could be due to reduced losses prior to the rotor inlet, resulting in more energy available for the rotor to extract from the flow.

Chapter 6: Re-optimization of Injection Parameters

This chapter presents the procedure and results for a re-optimization exercise conducted on the injection parameters. The re-optimization was conducted by using the rotor inlet velocity profiles obtained from the full stage vaned turbine simulations as inlet boundary conditions on a single passage domain. Optimization was conducted on a range of operating conditions at 30,000rpm and 42,000rpm. Results from the re-optimization show that at VR higher than the peak efficiency point, the injected flow has reduced the entropy generation near the tip gap by disrupting the formation of the tip leakage vortex. However, at VRs near or below the peak efficiency point there is no clear efficiency improvement from flow injection. The maximum efficiency obtained was 2.71 percentage points at operating point 30-3.

6.1 Investigation Procedure

The previous section showed that with a vaned turbine the suction side separation vortex was unable to form, thus, secondary flow injection from the shroud did not have a major impact in reducing the losses arising from the separation vortex. With the data available from the full stage turbine simulations, it is then possible to impose the rotor inlet flow field from the full stage simulations as the inlet boundary condition to a single passage model. This allows for a re-optimization of the injection parameters in order to investigate if other areas of flow improvement could be obtained through secondary flow injection. To expand the scope of investigation, optimization was conducted at 6 different operating points at two different turbine speeds, shown in Table 6.1. The operating points were chosen around the peak efficiency of that turbine speed. One of the points will be close to the peak efficiency point at each speed, one will have a mass flow below the peak efficiency point, and one will have a mass flow above the peak efficiency point. This provides an idea on the effectiveness of secondary flow injection at operating points below, at, and above the peak efficiency point. It also allows for a comparison on the optimized injection parameter values at each operating point. The optimization method was the same as that described in Chapter 3.4, but the range of parameters used was changed slightly from Table 3.3. The new range of parameters is found in Table 6.2.

Table 6.1: Single Passage Injection Re-Optimization Operating Points

Operating Point	Turbine Speed (rpm)	Mass flow rate (kg/s)
30-3	30,000	0.2043828
30-5	30,000	0.2673696
30-7	30,000	0.3736452
42-5	42,000	0.3528936
42-7	42,000	0.452226
42-9	42,000	0.5433204

Since the inlet boundary condition was obtained from the full stage turbine simulations, the rotor domain used was identical to the one used in the full stage simulations, and the inlet was not extended. This is shown in Figure 6.1, with the user-defined surfaces for turbine inlet and outlet measurements shown in black.

Table 6.2: Re-optimization Injection Parameter Ranges

Parameter	Lower Limit	Upper Limit
Slot Location (from blade tip LE)	4.0mm	17.0mm
Slot Width	1.0mm	6.0mm
Injection Velocity	10m/s	250m/s
Injection Yaw Angle	-80°	80°
Injection Pitch Angle	10°	89°

Figure 6.1: Re-optimization Simulation Domain, With User-defined Turbine Inlet and Outlet Surfaces

6.2 Results

Table 6.3 shows the results of the re-optimization simulations. The efficiency curves are also plotted against the baseline in Figure 6.2. The highest improvements in efficiency were obtained at points 30-3 and 42-5, with an efficiency improvement of 2.71 and 1.53 percentage points respectively. These are the two points that are below the peak efficiency point in terms of passage mass flow rate. This suggests that the benefits of secondary flow injection are greater at off-design operating points that are generally low pressure ratio and mass flow. The results also show that at points 30-3 and 42-5, the efficiency obtained through injection was around the same as the baseline peak efficiency. This shows the potential of injection in improving the range that the turbine can operate at high efficiencies.

Table 6.3: Re-Optimization Results

Operating Point	Optimized Parameters					Efficiency (%)	Baseline Efficiency (%)	Efficiency Change (% points)
	Slot Location (mm)	Slot Width (mm)	Injection Velocity (m/s)	Injection Yaw Angle (°)	Injection Pitch Angle (°)			
30-3	14.3	2.3	48	64	76	80.739	77.984	2.755
30-5	14.8	1.4	16	-77	89	80.783	80.283	0.50
30-7	8.2	3.1	10	79	85	77.199	76.656	0.543
42-5	15.4	1.8	51	21	89	81.784	80.250	1.534
42-7	12.8	3.2	10	-64	81	81.413	80.814	0.599
42-9	11.1	2.9	11	6	79	80.061	79.688	0.373

A detailed look into the flow field for all 6 operating points is shown in Figure 6.3 to Figure 6.8. The figures show contours of entropy generation, as well as the injection streamlines in red, and tip leakage streamlines in yellow. For operating point 30-3, it can be seen from Figure 6.3 that the injected flow has reduced the entropy generation near the tip gap by disrupting the formation of the tip leakage vortex. However, the mixing of the injected flow with the tip leakage flow has produced some entropy generation on the blade suction surface. A similar phenomenon could be found, but to a smaller extent, in operating point 42-5, shown in Figure 6.6. This could explain the larger improvements in efficiency at these two operating points.

For all the other operating points (30-5, 30-7, 42-7, 42-9), secondary flow injection did not show clear improvements in entropy generation near the blade tip. However, it could be seen that the mixing of the injected flow and the tip leakage flow has caused regions of high entropy generation in all these cases. Thus, it is unclear how the small efficiency improvements for these cases were achieved. It is suspected that this small increase in efficiency might simply be a numerical uncertainty arising from the location the turbine temperature and pressure, and mass flow measurements were taken (taking the values from a slightly different location will result in slightly different values, and these differences, especially temperature, can result in differences up to a scale of $10^{-1}\%$ in efficiency values). Thus, the efficiency benefits of injection at these points cannot be confirmed.

Figure 6.2: Re-optimized Injection Efficiency Comparison Against Baseline: Top) 30,000rpm, Bottom) 42,000rpm

Figure 6.3: Entropy Generation Contours and Streamlines for Leakage (Yellow) and Injection (Red) Flow, Point 30-3: Left) Baseline, Right) Re-optimized Injection

Figure 6.4: Entropy Generation Contours and Streamlines for Leakage (Yellow) and Injection (Red) Flow, Point 30-5: Left) Baseline, Right) Re-optimized Injection

Figure 6.5: Entropy Generation Contours and Streamlines for Leakage (Yellow) and Injection (Red) Flow, Point 30-7: Left) Baseline, Right) Re-optimized Injection

Figure 6.6: Entropy Generation Contours and Streamlines for Leakage (Yellow) and Injection (Red) Flow, Point 42-5: Left) Baseline, Right) Re-optimized Injection

Figure 6.7: Entropy Generation Contours and Streamlines for Leakage (Yellow) and Injection (Red) Flow, Point 42-7: Left) Baseline, Right) Re-optimized Injection

Figure 6.8: Entropy Generation Contours and Streamlines for Leakage (Yellow) and Injection (Red) Flow, Point 42-9: Left) Baseline, Right) Re-optimized Injection

The results also show some trends in the optimized injection parameters. Firstly, the injection slot location seems to move more upstream as the operating mass flow increases. This might be due to the onset of loss features (e.g. tip leakage flow and separation vortex, see Figure 4.2) occurring more upstream as the turbine mass flow increases. Also, the injection velocity is close to or at the lower bound value set for the optimization (see Table 6.2) for operating points at or above the peak efficiency point (30-5, 30-7, 42-7, 42-9). This seems to suggest that it might have been better not to

have any injection at all at these points. Indeed the efficiency values suggest this as well, as explained earlier.

Chapter 7: Conclusions and Future Work

This chapter presents the conclusions drawn from the work conducted in this thesis. The main conclusions drawn are that through CFD flow visualization, the main sources of loss in the mixed flow turbine investigated are the tip leakage loss and the loss due to the suction side separation vortex. Secondary flow is able to increase turbine efficiency at high VR by disrupting the formation of tip leakage vortex, but at lower VR efficiency improvement is limited. A greater potential for efficiency increase lies in vaneless turbines, where secondary flow injection is able to reduce losses arising from separated flow near the hub. Future work on this topic can be in the direction of engine level performance evaluations, studies of injection under unsteady turbine flow conditions, exploring other injection design variables, as well as evaluating injection flow performance on an actual vaneless turbine.

This thesis presents the numerical and experimental investigation of using secondary flow injection over the shroud of a mixed flow turbine to reduce aerodynamic losses and improve the turbine efficiency. The novelty of the work are:

1. Providing a detailed understanding of the flow structures within a mixed flow turbine at various off-design operating points.
2. Evaluating the feasibility of using secondary flow injection as a novel method of improving the efficiency of a mixed flow turbine used in automotive turbochargers.
3. Using optimization to find out the optimum injection parameters to be used.

The following sections will discuss the main conclusions, which are closely linked to the research objectives stated in Chapter 1. Recommendations will also be made with regards to the future work on this concept.

7.1 Conclusions

- Evaluate the feasibility in using secondary flow injection from the turbine casing to reduce losses within a mixed flow turbocharger turbine and improve efficiency.

An extensive amount of steady-state CFD simulations, validated by experimental data, has been conducted to assess if secondary flow injection over the turbine shroud can improve the flow field within a mixed flow turbine, reduce aerodynamic losses, and in turn increase turbine efficiency. The results showed that with a vaned turbine, it is difficult to achieve significant improvements in efficiency at operating points with mass flows at or above the peak efficiency point (lower VR). Detailed flow structure analysis showed that the injected flow tends to mix with the tip leakage flow, resulting in mixing losses that showed up as regions of high entropy generation. At operating points with mass flows lower than the peak efficiency point (higher VR), however, the CFD results show that secondary flow injection is able to disrupt the formation of the tip leakage vortex and reduce the losses arising from the vortex, leading to an efficiency improvement.

For a vaneless turbine, CFD results have shown that it is possible to significantly increase turbine efficiency over a range of operating points through secondary flow injection. Detailed flow analysis has shown that without nozzle vanes to guide the flow, a large separation vortex tends to form on the suction surface of the turbine, which is a significant source of loss in the turbine. With injection, the injected flow creates a nozzle effect which forces the passage flow to more closely follow the streamwise direction. This helps to weaken the formation of the separation vortex, and in turn reduce entropy generation arising from the vortex. As a result, an improvement in efficiency can be achieved. An interesting result with the vaneless turbine is that without injection, the vaneless turbine has a lower peak efficiency compared to a vaned turbine, but with injection, the vaneless turbine had a higher peak efficiency than the vaned turbine. The reason for this is suspected to be due to reduced losses prior to rotor inlet which improved the turbine torque produced, but further investigations are needed for a clear conclusion.

- Gain greater understanding of the flow structures inside a mixed flow turbine and how they interact with flow injection.

Detailed flow analysis of the mixed flow turbine over a range of different operating points was conducted and presented in this thesis. The flow visualizations using CFD indicated clearly the secondary flow structures within the turbine, such as the suction side separation vortex, as well as the leakage flow structures. Entropy generation contours were used to identify regions of loss, and these can be compared with the streamlines to identify loss mechanisms such as mixing of two flows. The entropy generation contours showed that the tip leakage vortex is a major source of entropy generation, which is consistent with results found in literature. The suction side separation vortex is a major source of loss as well.

As the source of injection is over the shroud of the turbine, the tip leakage flow will always be affected by injection, even when the injection mass flow rate is small. At some operating points, the injected flow will disrupt the formation of the tip leakage vortex and mitigate the losses arising from the vortex, but the mixing of the injected flow and the tip leakage vortex will always generate additional loss. Injection can also affect flow further down the passage, potentially able to weaken separations that occur

near the hub as well. This makes secondary flow injection potentially a powerful method for improving turbine performance.

7.2 Future Work

7.2.1 Engine Level Study

As a step further in the research the concept should be tested on an actual internal combustion engine. The first step to this can be an engine simulation using a 1-D simulation package such as GT-Power. Some preliminary work has already started in this area, but the difficulty lies in deciding where to obtain the injected fluid from. Modelling a turbine with injection partway over the rotor using a 1-D model is another challenge as well.

7.2.2 Unsteady Tests on Injection

For the current thesis, only steady-state turbine performance has been evaluated. However, a turbocharger turbine works under highly pulsating flow conditions. Thus, one of the next step in the research will be to conduct unsteady tests on the multi-injection turbine housing concept. Another concept that can be tested is the usage of pulsed injection flow.

7.2.3 Alternative Injection Designs

For the current thesis, only one injection opening design was evaluated (a single circumferential slot), although variations of the width and location of the slot were investigated. For future work, the design space can be greatly expanded. Multiple discrete holes, or multiple injection slots are some of the alternative designs that can be explored.

7.2.4 Injection on Vaneless Turbine

As seen from previous results, the multi-injection turbine housing concept seems to offer better performance improvements in a vaneless turbine. It will be interesting to evaluate this concept on an actual vaneless turbine to assess the benefits of flow injection.