Influence of Inlet Temperature Distortion on Turbine Heat Transfer

by

Tonghuo Shang

BEng, Beijing Institute of Aeronautics and Astronautics (1985)
S.M., Massachusetts Institute of Technology (1989)

SUBMITTED TO THE DEPARTMENT OF AERONAUTICS AND ASTRONAUTICS IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF DOCTOR OF PHILOSOPHY

at the

MASSACHUSETTS INSTITUTE OF TECHNOLOGY

February 1995

© 1995 Massachusetts Institute of Technology

All rights Reserved

Signature of Author

Dept. of Aeronautics and Astronautics, January 19, 1995

Certified by ________________

Prof. Alan H. Epstein
Thesis Supervisor, Professor of Aeronautics and Astronautics

Certified by ________________

Dr. Gerald R. Guenette, Principal Research Engineer

Certified by ________________

Prof. Jack L. Kerrebrock
R.C. MacLaurin Professor of Aeronautics and Astronautics

Certified by ________________

Prof. John Dugundji, Senior Lecturer/Professor Emeritus

Accepted by ________________

Prof. Harold Y. Wachman, Chairman, Departmental Graduate Committee
Influence of Inlet Temperature Distortion on Turbine Heat Transfer

by

Tonghuo Shang

Submitted to the Department of Aeronautics and Astronautics
on January 19, 1995 in partial fulfillment of the requirements for the
degree of Doctor of Philosophy

ABSTRACT

The influence of inlet temperature distortions typical of combustor exit flows on turbine heat transfer was investigated both experimentally and computationally, using a highly loaded transonic turbine as test article.

The experimental work was carried out at the MIT blowdown turbine facility. Heat flux was measured on rotor blade surface at three sections representing nominally tip, midspan and hub radii. Aerodynamic data was taken in conjuncture with heat flux measurements for data reduction and analysis.

The data shows that a 12% radial temperature distortion (RTDF) increases rotor blade heat transfer by 20-30%, except on the hub suction surface where a reduction of 20% in heat flux was observed. At midspan, the heat flux increase was caused by local inlet gas temperature rise. Tip and hub heat flux changes were not explained by local inlet gas temperature changes. Examination of heat flux distribution implies that the influence of secondary flows is important.

A 10% circumferential temperature distortion (OTDF) was found to have a small influence on blade heat transfer, confirmed by the experimental data and 3-D Euler calculations. Further computational investigation with higher levels of hot streak temperature distortion revealed three mechanisms by which an inlet temperature distortion can influence the rotor blade surface heat transfer. First, preferential migration of hot/cold fluid leads to higher time-averaged local surface temperature nonuniformity between the pressure and suction surfaces. This effect scales nonlinearly with temperature distortion amplitude. Second, the influence of buoyancy drives high temperature fluid toward lower radii and the rotor platform. This radial displacement is proportional to hot streak temperature and inversely proportional to local flow coefficient. Third, unsteady blade row interaction causes the stator exit flow angle to fluctuate and the hot streak to wobble at the rotor blade passing frequency. Viewed in the rotor relative frame, the hot streak passes the rotor at varying speeds, leading to an azimuthal variation in the time-averaged rotor inlet temperature. As a result, rotor leading edge temperature is different from the azimuthal mean. This influence is linearly proportional to hot streak amplitude, and is a function of stator-rotor pitch ratio. Implications of the above to turbine design are noted.

Thesis Supervisor: Dr. Alan H. Epstein
Title: Professor of Aeronautics and Astronautics
Acknowledgments

The road to this degree has been long and challenging. Many people have helped make the journey a successful and rewarding one. I would like to take this opportunity to thank all those who have made this MIT experience worthwhile.

First I would like to express my respect and gratitude to Prof. Alan H. Epstein and Dr. Gerald R. Guenette. Their guidance, encouragement and friendship have been crucial to the success of this project. I would also like to thank other members of my thesis committee, Prof. Jack L. Kerrebrock and Prof. John Dugundji for their good suggestions during this work. Special thanks go to Prof. Edward M. Greitzer and Dr. Choon S. Tan for donating their valuable time to read my thesis and for offering constructive comments.

I would like to thank the other students I have worked with on the experimental work. Martin Graf and Kendra Hardie for their skill in instrumentation, David Sujudi for his expertise in A/D systems and Emmanuel D’Hoop for taking tunnel turbulence data. I also wish to thank Dr. Andre Saxer for setting up the CFD code.

My gratitude also goes to Holly Rathbun for her help in acquisition and purchasing, Diana Park for technical illustration and Robin Courchesne for making sure everything is all right. The expedient assistance by Bill Ames, Jimmy Letendre and Viktor Dubrowski are especially acknowledged.

I am deeply indebted to my wife Wenchi for her patience, support and sacrifice during all these years. She postponed her own studies and brought up our two lovely children, Si-Si and Sen-Sen. The little ones were constant sources of joy. Finally I dedicate this thesis to my parents, whose love and care have provided spiritual drive for my pursuit to the maximum of my abilities.

This research was supported by:

US Air Force Office of Scientific Research
NASA Lewis Research Center
Rolls Royce Inc., and
SNECMA
Table of Contents

Title Page
Abstract
Acknowledgments
Table of Contents
List of Figures
List of Tables

1. Introduction ................................................................................................. 17
   1.1. Motivation ............................................................................................. 17
   1.2. Literature Review ................................................................................ 20
   1.3. Objective and Approach ..................................................................... 29
   1.4. Thesis Outline ..................................................................................... 29

2. Experimental Apparatus ............................................................................. 37
   2.1. Blowdown Turbine Facility ................................................................. 37
   2.2. Radial Temperature Distortion Generator ........................................ 41
   2.3. Hot Gas Injection .................................................................................. 44
   2.4. Turbulence Grid ................................................................................... 45
   2.5. Circumferential Rake Translators ...................................................... 46
   2.6. Instrumentation and Data Acquisition Systems .................................. 50

3. Numerical Procedure .................................................................................. 75

4. Data Reduction and Experimental Results ............................................. 81
   4.1. Test Matrix and Tunnel Conditions .................................................... 81
   4.2. Heat Flux Data Reduction and Uncertainty Estimate ....................... 89
   4.3. Heat Transfer Measurements .............................................................. 97
   4.4. Summary ............................................................................................. 102

5. Heat Transfer Data Analysis and Comparison with CFD Results .......... 123
   5.1. Introductory Remarks ......................................................................... 123
   5.2. Rotor Surface Heat Transfer with Radial Temperature Distortion ........ 127
   5.3. Rotor Surface Heat Transfer with Circumferential Distortions .......... 132

6. Computational Analysis of the Redistribution of Hot Streak in A Transonic Turbine Stage ......................................................... 157
   6.1. Approach ............................................................................................ 158
   6.2. Description of Flowfield and Hot Streak Migration in the Rotor ........ 159
   6.3. Buoyancy Effects ............................................................................... 160
List of Figures

Fig. 1.1. Trend of turbine inlet temperature with time. From [30]
Fig. 1.2. Current prediction methods under estimate rotor pressure surface heat transfer. Symbols represent different engine companies [19].
Fig. 1.3: Combustor exit dynamic temperature signals from an engine test. Average burner exit temperature 1200°C. From [16].
Fig. 1.4: Typical radial and circumferential temperature distribution for an annular burner at 2500°F. From [53].
Fig. 1.5: Measured CO2 concentration contours with and without introduction of temperature distortion. [5]
Fig. 1.6: Predicted mid-span rotor surface relative total temperature distribution using 2-D CFD codes. From [14].
Fig. 1.7: Predicted (3-D Euler and N-S Codes) and experimental time averaged surface temperature for rotor mid-span section [14].
Fig. 2.1: Schematic of the MIT blowdown turbine facility.
Fig. 2.2: Cross-sectional view of the Blowdown Turbine facility internal flowpath, upstream of turbine stage.
Fig. 2.3: Cross-sectional view of the internal flowpath, downstream of turbine stage.
Fig. 2.4: Blowdown Turbine’s radial temperature distortion generator.
Fig. 2.5: Picture of the RTDFG honeycomb matrix.
Fig. 2.6: Schematic of the hot gas injection system.
Fig. 2.7: Turbine inlet flow geometry and injector locations. mechanical struts divided flowfield into 3 equal sectors.
Fig. 2.8: Drawing of the turbulence grid. Three identical pieces can be installed.
Fig. 2.9: Upstream rake translator. The struts in the cutaway view are shown out of position.
Fig. 2.10: Downstream rake translator configured for rotor exit flow measurement.
Fig. 2.11: Downstream translator electrical ribbon loop containment. The ribbon indicated consists three electrical ribbons sandwiched between 0.003" stainless shims for stiffness.
Fig. 2.12: Downstream rake translator configured for NGV exit flow survey.
Fig. 2.13: Functional diagram of the translator position control.
Fig. 2.14: Upstream translator speed and position trajectory for a typical blowdown test. Angular position defined in Fig. 2.7.
Fig. 2.15: Aero instrumentation layout in blowdown turbine. Nomenclature in () indicating probe designation.
Fig. 2.16: NGV exit static tap positions. Drawing not to scale.
Fig. 2.17: Multi-layer heat flux gauge. Drawing not to scale.
Fig. 2.18: Heat flux gauge locations around the rotor blade.
Fig. 2.19: Rotor blade heat flux gauge sensor location and status.
Fig. 3.1: Computational grid for the ACE stage, axial radial plane.
Fig. 3.2: Computational grid in the pitchwise direction.
Fig. 4.1: Upstream total temperature field for uniform conditions.
Fig. 4.2: Total temperature contours measured by the inlet translating rake at the injector exit plane.
Fig. 4.3: NGV exit hot spot total temperature contours (domain of 3 NGV passages).
Fig. 4.4: NGV exit temperature distribution downstream of a hot spot measured at three spanwise locations.
Fig. 4.5: Radial variation of hot spot center line temperature measured upstream and downstream of NGV.
Fig. 4.6: Inlet total temperature contours for the radial distortion test, T169 sector B.
Fig. 4.7: Radial temperature variation for uniform and radial distortion tests.
Fig. 4.8: Radial temperature profile for full stage and NGV only tests.
Fig. 4.9: NGV exit radial temperature distribution for uniform and radial distortion tests.
Fig. 4.10: Turbulence power spectral density generated by grid. Measurement with dual hot wire probe.
Fig. 4.11: Comparison of current data and previous measurements of rotor Midspan heat transfer.
Fig. 4.12: Midspan heat transfer with uniform inlet temperature - Data Repeatability.
Fig. 4.13: Tip heat transfer with uniform inlet temperature - Data repeatability.
Fig. 4.14: Hub heat transfer with uniform inlet temperature - Data repeatability.
Fig. 4.15: Repeatability of time resolved heat transfer. Midspan sensor #11.
Fig. 4.16: Repeatability of time resolved heat transfer data. Midspan sensor #14.
Fig. 4.17: Midspan heat transfer at three 120° sectors - Tunnel uniformity in θ.
Fig. 4.18: Tip heat transfer at three 120° annular sectors - Tunnel uniformity in θ.
Fig. 4.19: Hub heat transfer at three 120° annular sectors. Tunnel uniformity in θ.
Fig. 4.20: Rotor surface heat load vs. chord and span. uniform inlet temperature. Data taken from test T173 sector B.
Fig. 4.21: Effect of grid turbulence on rotor heat transfer - Midspan.
Fig. 4.22: Effect of grid turbulence on rotor heat transfer - Tip section.
Fig. 4.23: Effects of grid turbulence on rotor heat transfer - Hub section.
Fig. 4.24: Pressure side comparison of grid turbulence influence.
Fig. 4.25: Suction side comparison of θTDF heating influence.
Fig. 4.26: Pressure side comparison of θTDF heating influence.
Fig. 4.27: Influence of θTDF on rotor heat transfer - Midspan.
Fig. 4.28: Influence of θTDF on rotor heat transfer - Tip section.
Fig. 4.29: Influence of θTDF on rotor heat transfer - Hub section.
Fig. 4.30: Influence of RTDF on rotor heat transfer - Midspan.
Fig. 4.31: Influence of RTDF on rotor heat transfer - Tip section.
Fig. 4.32: Influence of RTDF on rotor heat transfer - Hub section.
Fig. 5.1: Velocity triangles at the NGV rotor interface.
Fig. 5.2: Cascade heat transfer data by Ashworth et al [4].
Fig. 5.3: Midspan heat transfer distribution at design and -10° incidence.
Fig. 5.4: Midspan Nusselt number comparison between 12% RTDF and uniform inflow conditions.
Fig. 5.5: Nusselt number comparison between 12% RTDF and uniform inflow conditions. Tip section (79% span).
Fig. 5.6: Nusselt number comparison between 12% RTDF and uniform inflow conditions at nominal Hub section (16% leading edge span).

Fig. 5.7: Chordwise-integrated pressure surface Nusselt number at three blade sections of hub (16%), midspan (46%) and tip (79%). Reference gas temperature from 2D streamline method.

Fig. 5.8: Chordwise-integrated suction surface Nusselt number at three blade sections of hub (16%), midspan (46%) and tip (79%). Reference temperature from 2D streamline method.

Fig. 5.9: Parabolic NGV inlet temperature profile fit from data.

Fig. 5.10: Snapshot of unsteady static pressure in the blade-to-blade rotor passage at midspan. There is a difference in contour intervals.

Fig. 5.11: CFD Calculated rotor inlet relative flow angle with and without radial distortions.

Fig. 5.12: CFD calculated rotor inlet relative total pressure as function of span.

Fig. 5.13: Rotor relative total temperature contours in the blade-to-blade passage. Hot midspan gas migrates to the pressure surface and spreads out radially toward endwalls. Temperature normalized by NGV inlet average.

Fig. 5.14: Rotor surface streamlines with uniform inflow conditions.

Fig. 5.15: Rotor surface streamlines with 12% RTDF.

Fig. 5.16: Rotor surface total temperature contours from CFD results with uniform inlet temperature. Temperature normalized by NGV inlet average.

Fig. 5.17: Rotor surface relative total temperature contours with 12% RTDF. Temperature normalized by NGV inlet average.

Fig. 5.18: Midspan Nusselt number comparison between 12% RTDF and uniform inflow conditions, using local gas temperature from CFD code.

Fig. 5.19: Nusselt number comparison between 12% RTDF and uniform inflow conditions. Tip section (79% span).

Fig. 5.20: Nusselt number comparison between 12% RTDF and uniform inflow conditions, using CFD calculated surface temperature as reference. Hub section (16% span).

Fig. 5.21: Reproduced from ref [28]. Temperature contours in axial cuts of a rotor passage, showing tip leakage flow draws midspan fluid to tip region.

Fig. 5.22: Tip section pressure surface Nusselt number comparison between 12% RTDF and uniform conditions. Midspan gas temperature used to normalize tip section heat flux.

Fig. 5.23: Surface flow visualization of the same rotor geometry in a transonic cascade wind tunnel [4], Showing endwall secondary flows.

Fig. 5.24: NGV passage ensemble-averaged unsteady heat flux with and without radial temperature distortion. Midspan.

Fig. 5.25: NGV passage ensemble-averaged unsteady heat flux with and without radial temperature distortion. Tip section.

Fig. 5.26: NGV passage ensemble-averaged unsteady heat flux with and without radial temperature distortion. Hub section.

Fig. 5.27: Midspan rotor Nusselt number comparison between 10% 0TDF and uniform case. 2D streamline method.
Fig. 5.28: Tip section rotor Nusselt number comparison between 10% OTDF and uniform case. 2D streamline method.
Fig. 5.29: Hub section rotor Nusselt number comparison between 10% OTDF and uniform case. 2D streamline method.
Fig. 5.30: Heat flux changes due to blade surface temperature variation from leading edge values. CFD calculation with a 10% OTDF.
Fig. 5.31: Midspan Nusselt number comparison between uniform and 10% OTDF. Surface temperature from CFD calculation used as reference temperature.
Fig. 5.32: Tip section Nusselt number comparison between uniform and 10% OTDF. Surface temperature from CFD calculation used as reference temperature.
Fig. 5.33: Hub section Nusselt number comparison between uniform and 10% OTDF. Surface temperature from CFD calculation used as reference temperature.
Fig. 5.34: Ensemble-averaged unsteady heat flux data with and without circumferential temperature distortion. Midspan.
Fig. 5.35: Ensemble-averaged unsteady heat flux data with and without circumferential temperature distortion. Tip section.
Fig. 5.36: Ensemble-averaged unsteady heat flux data with and without circumferential temperature distortion. Hub section.
Fig. 6.1: NGV inlet temperature distortion in the form of a circular hot streak.
Fig. 6.2: Snapshot of flowfield showing convection of hot streak through turbine stage.
Fig. 6.3: Time averaged total temperature distribution on rotor's pressure and suction surfaces. Hot streak temperature ratio of 1.8. Temperature normalized by midspan leading edge value with contour intervals of 0.02.
Fig. 6.4: Snapshot of rotor exit flow field (looking upstream) showing position of hot streak.
Fig. 6.5: Time-averaged chordwise velocity distribution on rotor pressure surface. Extracted from midspan under uniform NGV inlet conditions.
Fig. 6.6: Simple model predicted and CFD calculated hot streak core trajectory on rotor pressure surface. Mean surface streamline from uniform calculation.
Fig. 6.7: Time-averaged rotor relative total temperature distribution on rotor platform. Temperature normalized by averaged value at hub inlet.
Fig. 6.8: Surface gas temperature distribution and resulted heat flux change, with hot streak to mean flow temperature ratios of 1.4 and 1.8.
Fig. 6.9: Time-averaged rotor relative total temperature at rotor's leading edge plane. Contour intervals are about 2% of averaged rotor inlet relative temperature.
Fig. 6.10: a) Time-averaged rotor relative total temperature at axial station 20% rotor chord upstream of leading edge. b) Midspan pitchwise variation from a).
Fig. 6.11: a) Hot Streak calculation using a 2-D Euler solver based on midspan flow conditions of 3-D Calculation, Fig. 6.10. b) Pitchwise variation of time-averaged rotor relative total temperature 20% rotor chord upstream of leading edge.
Fig. 6.12: a) Rotor only calculation simulating unsteady hot streak. Inlet conditions based on 2-D stator-rotor calculation, Fig. 6.11. b) Pitchwise variation of time-averaged rotor relative total temperature 20% rotor chord upstream of leading edge.
Fig. 6.13: NGV exit flow angle history.
Fig. 6.14: Two extreme positions of the hot streak showing hot streak wobble. Entropy used as marker.

Fig. 6.15: Time history of rotor relative total temperature at two pitchwise positions with maximum and minimum time-averaged values. Temperature normalized by freestream value. Traces are time shifted for comparison.

Fig. 6.16: Time history of rotor relative total temperature at two pitchwise positions. Hot streak to mean flow temperature ratio of 1.4.

Fig. 6.17: Peak-to-peak magnitude of pitchwise variation of time-averaged rotor relative total temperature at midspan with hot streak temperatures of 1.4 and 1.8. Temperature normalized by circumferential average.

Fig. 6.18: Time-averaged rotor relative total temperature at leading edge plane. Calculation with stator rotor blade count of 3:4 and 3:5. Hot streak temperature ratio of 1.8.

Fig. 6.19: Planar cuts through rotor flow field showing time-averaged relative total temperature. Contour intervals of 2% rotor inlet averaged temperature.

Fig. A.1: Schematic of the injector mass flow calibration arrangement.

Fig. A.2: Orifice plate mass flow characteristics. Corrected mass flow vs. pressure ratio.

Fig. A.3: Data scatter of injector mass flow using calibrated orifice plate as flowmeter.

Fig. C.1: Two Dimensional Heat Conduction in the Honeycomb Matrix.

Fig. C.2: 1D Heat Conduction model.

Fig. C.3: Domain of 2D Heat Conduction Model.

Fig. C.4: 1D unsteady heat conduction model solutions.

Fig. C.5: Fit between data and 1D conduction model. Symbols are data taken from tip to hub at five locations.

Fig. C.6: Fit between data and 2D heat conduction model. Symbols are matrix temperatures at various radial and circumferential locations.

Fig. C.7: Model predicted temperature distribution in sector A of the matrix heat exchanger with different heating times.
List of Tables

Table 1-1 : Geometric Parameters and Test Conditions for the Hot streak experiment by Butler, et al. [5]
Table 2-1: MIT Blowdown Turbine Scaling.
Table 2-2: Design Parameters of ACE stage.
Table 2-3: Radial Positions of the Inlet Translating T/C sensors.
Table 2-4: Radial Positions of the Inlet Total Pressure Rake Sensors
Table 2-5: Radial Positions of the Downstream Translator Rake Sensors
Table 2-6: Heat Flux Gauge Locations.
Table 4-1: Full Stage Test Matrix
Table 4-2: NGV Only Test Matrix.
Table 4-3: Freestream and Injector flow properties, T171.
Table 4-4: NGV and Rotor Exit Mach Numbers.
Table 4-5: Stage Test Tunnel Conditions.
Table 4-6: Test Conditions for Tests T174 and T52.
Table 5-1: Nusselt number difference between radial distortion and uniform tests.
Table A-1: Calibrated effective flow area of orifice plates used in Eq A.6. (α = 0.346)
Chapter 1.
Introduction

1.1. Motivation

One of the main contributing factors to higher specific core power and lower fuel consumption of gas turbine engines is increased turbine inlet gas temperature [32]. Over the last forty years significant increase in this temperature has been achieved, as illustrated in Fig. 1.1. While the development of high temperature materials has contributed steadily over time, the introduction of turbine cooling in the 1960’s has considerably accelerated the improvement. Modern state of the art commercial transport engines have turbine inlet temperatures of over 1700°K, and it is higher for military engines with shorter turbine life. This temperature level, however, is much less than the stoichiometric combustion temperature of 2500°K. So that there is still a substantial margin for improvement.

With the gas temperature much higher than the turbine material can tolerate, cooling must be applied to keep the metal temperature close to uniform and within allowable limits, so as to avoid thermal stress and surface oxidation. For this purpose, elaborate cooling schemes have been devised and distributed around the airfoil surface, using relatively cool air extracted from the compressor. Many different cooling schemes can be implemented simultaneously, such as internal convective, impingement, and film cooling. Internal cooling (cooling air flows inside the hollow turbine airfoil) aims at absorbing the heat conducted into the blade without affecting the external fluid dynamics. Film cooling, on the other hand, bleeds coolant air through holes or slots on the airfoil
surface, forming a thin “film” barrier between the hot surrounding gas and the blade. Blade surface heat transfer varies about the airfoil, from leading edge to trailing edge, and from pressure surface to suction surface, so that cooling design must be tailored to account for this heat load distribution. Localized inadequate or excessive cooling causes temperature gradients, leading to thermal stress. High pressure first stage turbines often uses as much as 15-20% of compressor core mass flow as cooling air, with considerable penalties on turbine aerodynamics and engine performance [53]. In this context, it is highly desirable to optimize the cooling distribution and effectiveness and minimize the cooling flow requirement.

Detailed cooling design requires accurate determination of surface heat load distribution on the turbine blade\(^1\). Over the past forty years, the prediction capability has improved considerably. High frequency response heat flux sensors allowed accurate heat transfer measurement on the blade surface [18]. Supported by experimental data and increased computer power, calculational tools such as N-S solvers in 2-D as well as 3-D are becoming widely available. Abhari [1,2] compared such computational results to experimental data, taken both from linear cascades and fully simulated rotating rigs. Good agreement was found among them.

The experience in a real engine environment, however, is much different. Turbine blade heat transfer levels are considerably higher, as much as 50 to 100% higher than predicted, see Fig. 1.2. This difference in heat transfer implies that the flow characteristics in an engine are different than that modeled in test rigs and computation routines. Where does that difference lie? Since data from cascades and stages with rotating blade rows agree [2], unsteadiness and centrifugal effects do not appear to be controlling factors.

---

\(^1\)To demonstrate the degree of accuracy required, we take the test turbine used in this study, the ACE stage, as an example. The designed turbine inlet and metal temperatures are 1780°K and 1118°K respectively. For a 5% under-estimation of heat transfer coefficient, the average operating metal temperature would be 33°K higher than design (that is 5% of the gas-to-metal temperature difference, 662°K). This level of metal temperature variation means a 50% reduction in turbine life.
The difference might lie in the inlet flow condition that is not simulated in a rig environment. In an engine, the turbine is positioned downstream of the combustion chamber, where air and fuel mix in swirling flows and chemical reaction occurs. There is also the interaction between the burned hot gas (near stoichiometric temperatures) and the cooler gas from the compressor discharge. As a result, the flow at the combustor exit is highly fluctuating and nonuniform. Apart from the high velocity turbulence, there are large magnitude spatial and temporal temperature variations as well. Measurements using compensated thermocouple probes have shown that the unsteady temperature at the combustor exit can vary as much as \( \pm 500^\circ K \) about an average combustor exit gas temperature of \( 1200^\circ K \) at intermediate power, see Fig. 1.3. Time averaged gas temperature varies in both radial and circumferential directions, due to both design intent and incomplete mixing of hot and cold gas streams. Circumferential temperature variations are related to the finite number of fuel injectors installed, with the highest temperature downstream of each injector (hot spot or hot streak). Nozzle guide vanes (NGV's or stators) are normally positioned such that the fuel injectors are aligned with the center of NGV passages. The circumferentially averaged radial variation, with temperatures lower near the hub and tip and higher in the middle, is a direct result of the cooling applied to the combustor liner. While circumferential temperature uniformity is one of the criteria of good combustors, the radial variation is desirable for the turbine, with lower temperatures needed at the hub because of the high centrifugal stress at the blade root.

References [30,53] show some typical temperature variations of combustor exit flow, with one such example given in Fig. 1.4. Hennecke[30] used the following terms to quantify this temperature nonuniformity: OTDF (overall temperature distribution factor) and RTDF (radial temperature distribution factor) defined as

\[
\text{OTDF} = \frac{T_{\text{max}} - T_{\text{avg}}}{\Delta T_{\text{combustor}}}
\]

Eq 1.1
and

\[ RTDF = \frac{T_{\text{max},r} - T_{\text{avg}}}{\Delta T_{\text{combustor}}} \]  

Eq 1.2

where \( T_{\text{avg}} \) is the spatially averaged combustor exit gas temperature, and \( \Delta T_{\text{combustor}} \) the temperature rise from combustor inlet to exit. \( T_{\text{max}} \) is the maximum combustor exit temperature and \( T_{\text{max},r} \) is the maximum temperature of the circumferentially averaged radial profile. The data included in reference [30] showed a 30% OTDF and 8% RTDF.1

NGV heat transfer is clearly affected by this temperature nonuniformity. The question of how turbine rotor responds to it is not so obvious and is the central issue of this study. The influence of momentum turbulence, typical of combustor exit flows both in intensity and spectrum distribution, will also be addressed.

1.2. Literature Review

As early as in 1947, Munk and Prim [38] demonstrated that for the steady inviscid flow of perfect gas with a fixed geometry, the streamline patterns are the same regardless of the total temperature variations, provided that total pressure distribution is the same. Under these conditions, the local velocity vector is simply proportional to the square root of the local total temperature, and nondimensional flow parameters such as Mach number are unchanged. This is often referred to as the Munk and Prim substitution principle. An extension of this principle to account for viscous stresses and heat transfer (both are present in realistic flows) was proposed by Greitzer et al [23]. This principle applies directly to the flow conditions of the turbine stator (nozzle guide vane or NGV),

---

1If the maximum temperature is at stoichiometric levels (2300-2500 K), and the averaged turbine inlet gas temperature is 1780 K, then the maximum OTDF would be between 50 to 70%.
where inlet total pressure variations are small due to the low Mach number flow in the combustion chamber. The total temperature variations are simply convected along the streamlines from inlet to exit.

The secondary flow theory of Hawthorne [29] and Lakshminarayana [35] also concluded that the growth of streamwise vorticity only depends on the gradient of total pressure, and total temperature gradients alone do not generate secondary flows in the stator. Because of the total pressure gradient in the rotor relative frame, temperature distortion introduces secondary flows in the rotor passage. Using the expressions relating the growth of streamwise vorticity to the temperature gradient, Lakshminarayana [35] analyzed the thermally driven secondary flows (rotation of isothermal surfaces) due to a radially varying temperature distribution, and showed the need for inclusion of the secondary flow effects in blade heat transfer calculation, cooling air requirement and material selection. Recently Graf [22] applied Hawthorne's secondary flow theory to the radial temperature distortion problem.

Perhaps the most cited experimental data on the effect of temperature distortion on turbine durability were that taken at the low speed rotating rig (LSRR) at UTRC by Butler, et al [5]. In his experiment, one circular hot streak was injected upstream of the NGV, at a location of 40% span from the hub, in the middle of one NGV passage. The size of the hot streak was 26% of NGV mid-span pitch. The total pressure of the injected flow was the same as that of the freestream. The injected flow was seeded with 15% CO$_2$ and the concentration levels in the flowfield were measured to track the flow trajectory of the hot streak. Measurements in the absolute as well as in the rotor relative frames were taken. In the absolute frame, concentration levels upstream (behind the injection point) and downstream of the NGV were measured by traversing rakes. In the rotor relative frame, radial rakes at the rotor leading and rotor surface static pressure taps were used to draw gas samples for CO$_2$ concentration analysis. Data taken in the rotor relative frame
were therefore time averaged values. Table 1-1 summarizes the turbine geometry and the
test condition as was given in [5].

Tests with and without temperature distortions were carried out to ascertain the
effects due to temperature distortion. For the distorted case, the hot streak temperature
was set to be twice that of the free stream. The NGV exit concentration measurements
indicated that the stator flow field is unaffected by the introduction of hot streak, as
predicted by the Munk and Prim substitution principle. However, the rotor flowfield is
significantly affected by the temperature distortion, shown here in Fig. 1.5, which causes
radial redistribution of hot gas from the mid span region and collection of hot gas on the
pressure surface. Two different physical mechanisms are identified as the cause: 1)
secondary flows due to the increase of inflow radial gradients in the rotor relative frame,
and 2) the preferential migration of hot gas due to the difference in inlet relative flow
angles into the rotor, a similar mechanism as described by Kerrebrock and Mikolajczak
[33] in compressors. The expanding and shrinking patterns on pressure and suction
surfaces with *uniform* inlet temperatures were caused by the endwall secondary flow
vortices. The radially outward flow between 50% span and tip on the pressure side are
related to the relative eddy discussed by Dring and Joslyn [15].
Table 1-1: Geometric Parameters and Test Conditions for the Hot streak experiment by Butler, et al. [5]

<table>
<thead>
<tr>
<th></th>
<th>NGV</th>
<th>ROTOR</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of airfoil</td>
<td>22</td>
<td>28</td>
</tr>
<tr>
<td>Axial chord (Meter)</td>
<td>0.151</td>
<td>0.161</td>
</tr>
<tr>
<td>Aspect ratio</td>
<td>1.01</td>
<td>0.946</td>
</tr>
<tr>
<td>Mid span inlet metal angle,</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>90°</td>
<td>42°</td>
</tr>
<tr>
<td>Mid span exit metal angle,</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>21°</td>
<td>26°</td>
</tr>
<tr>
<td>Inter-airfoil gap (% Chord)</td>
<td>65%</td>
<td></td>
</tr>
<tr>
<td>Flow coefficient, Cx/U</td>
<td>0.68</td>
<td></td>
</tr>
<tr>
<td>NGV inlet total temp., freestream</td>
<td>530°R</td>
<td></td>
</tr>
<tr>
<td>Injected flow temp.</td>
<td>1060°R</td>
<td></td>
</tr>
</tbody>
</table>

Tests with a hot streak to freestream temperature ratio of 1.2 were also done on the same rig and geometry, with a slightly different flow coefficient of 0.78. The results were subsequently used in the numerical investigations of Dorney et al [12].

Driven by the need to accurately predict the temperature redistribution with better understanding of the flow mechanisms and provide design guidance, a number of numerical investigations were carried out and compared to the Butler data. Rai and Dring [40] used a time-accurate 2-D Navier-Stokes code to simulate flow of a one-stator-one-rotor blade count of the LSRR turbine with an enlarged rotor. The blade row gap was reduced from 65% to 15%. A hot streak size equal to one fourth of the mid-span NGV pitch and temperature ratio of 1.2 (instead of 2.0) was implemented into the NGV inlet boundary specification. The calculated result showed qualitative agreement with experimental data, while quantitatively under-predicting the pressure surface temperature. Krouthen and Giles [34] used a coupled 2-D unsteady Euler/N-S scheme in which the N-S thin shear layer equations are solved in the boundary layer region around the airfoil while
Euler equations are used to simulate the flow in the passage. The calculation was performed on a single blade row where a periodic hot streak with twice the freestream temperature was prescribed as the boundary condition into the rotor. Effects due to flow coefficient and boundary layer states (fully turbulent or transitional) are examined. The results are similar to that reported by Rai and Dring [40]. Using the 2-D unsteady N-S solver developed by Rai, Dorney et al [14] examined the effect of blade number count (pitch ratio) on the temperature distribution by scaling the stator (rather than the rotor as Rai did). Configurations of 3-stator-4-rotor and 1-stator-1-rotor were examined with a stator-rotor gap of 15%. Again a streak to freestream temperature ratio of 1.2 was used to avoid numerical instability. The conclusions were that the blade count ratio has little effect on predicted time-averaged surface pressure and temperature distributions, but a substantial effect on the unsteady flow characteristics. Again quantitative agreement with data was lacking. It will be shown in Chapter 6 that blade count ratio is important in determining time-averaged surface temperature. Therefore the conclusions drawn in [14] regarding to blade count ratio is not generally correct.

Dorney et al later compiled a report [12] which included most of the published numerical studies [11, 13, 14]. Fig. 1.6, duplicated from [14], summarized all the two dimensional CFD predictions and compared to the experimental data. Data with a temperature ratio of 1.2 is also included as test case CHS2. A nondimensional surface temperature coefficient, relating to the concentration measurements are used as the following

$$C_T = \frac{T - T_1}{T_{avg rl e} - T_1} \approx \frac{CO_2 - CO_{2amb}}{CO_{2avg rl e} - CO_{2amb}}$$

Eq 1.3

where $T$ is the local time averaged temperature on the airfoil surface, $T_{avg rl e}$ is the mid-span time averaged temperature at the rotor leading edge, $CO_2$ is the local time
averaged CO₂ concentration, \( C_{O_{2\text{avg, rle}}} \) is the mid-span time averaged CO₂ concentration at the rotor leading edge, and \( T_I \) and \( C_{O_{2\text{amb}}} \) are the temperature and CO₂ concentration of free stream gas outside the hot streak. This definition inherently assumes the analogy between mass and heat transfer. Examination of Fig. 1.6 reveals that while the calculations do in general predict trends, the quantitative agreement is poor. The differences among the calculations and data are the same order of magnitude as the effects of interest. This inadequacy of 2-D CFD codes in predicting the temperature redistribution problem may be due to many factors. The main reason might be that the flow is inherently three dimensional, while a two dimensional calculation automatically assumes uniformity along spanwise direction.

Two dimensional hot streak was tested using the same facility with slight variations in the hardware geometry [51,13]. A rectangular hot jet with radially uniform circumferentially varying temperatures was injected upstream of the stator. The hot gas trajectory was measured through CO₂ concentration on the rotor surface. The experimental data indicates the hot gas accumulation on the pressure surface. Dorney et al [13] calculated the flowfield used the 2-D N-S procedure as in [14]. The numerical results agreed with the experimental measurements fairly well. It was further pointed out that the discrepancies were probably due to the gas sampling techniques in measuring the CO₂ concentration. Gas samples from well beyond the boundary layer may have been drawn into the sample analyzer. Therefore the surface CO₂ measurements are some average over the thickness of the boundary layer. When the surface temperature \( T \) used in coefficient \( C_T \) as in Eq 1.3 was modified to take the averaged total temperature across the boundary layer, excellent agreement existed between the calculation and the experiment. It should be pointed out that the calculation was performed on a one-stator one-rotor configuration and streak to freestream temperature of 1.2 was used.

Takahashi and Ni [54,55] used 2-D and 3-D, steady and unsteady Euler Codes in
their analysis. Surface shear modeling was used to account for viscous effects on the solid surfaces. It was concluded that 3-D unsteady calculations are necessary to capture all the physics related to a circular hot streak, i.e. the secondary flow effects and the unsteady segregation of hot and cold fluid. The predicted surface temperature distribution is qualitatively similar to the experimental data of Butler et al [5], although quantitatively under-predicted on the pressure surface. Dorney et al [14] performed 3-D N-S computations and showed strong effects due to end wall secondary flows, consistent with experimental observations, see Fig. 1.5. The predicted results from these two studies are shown in Fig. 1.7, from reference [14]. Again the quantitative agreement is poor.

More recent experimental studies were carried out by Roback and Dring [43]. The influence of hot and cold streaks, and NGV trailing edge coolant injection on rotor surface temperature were experimentally determined. Again the technique was using CO2 as a tracer gas of the hot (or cold) fluid. A number of temperature ratios and cooling injection velocity ratios were parametrically examined. Streak position relative the stator vane was varied from mid-pitch injection to leading edge injection (on stator streak). Experimental results indicated that hot streak fluids tend to accumulate on pressure surface while cold streak tends to collect on the suction side of the rotor blade. Also the suction surface accumulation of cold gas (cold streak or NGV coolant) is retarded by the end wall secondary vortices, which tend to bring endwall fluid to midspan on the suction surface and sweep midspan fluid to the tip and hub on the pressure surface. Another finding was that when both hot/cold streaks and NGV trailing edge cooling are present, the net influence is the sum of each individual contribution.

Most recently, Sharma et al [50] discussed the use of advanced numerical techniques in turbine design process, including the simulation of combustor generated hot streak influence on turbine heat load. It was shown that steady state codes, with circumferential averaging at blade row interface boundaries, are insufficient to describe
flows with circumferential flow variations like that of a hot streak. An "Average-Period" method, which solves the time averaged equation set with unsteady effects simulated as "apparent stresses," produced identical time averaged solution as that of unsteady calculations, implying significant reduction in computer resource requirements. The apparent stress, varying in three dimensions, was obtained from a time unsteady Euler solution. Because viscous regions also contribute towards establishing heat loads, and multistage, unsteady viscous codes were not yet practical in the iterative design process, it was proposed to utilize the "Average-Period" approach in steady 3-D viscous codes. Effects of periodic unsteadiness can be accounted for through apparent stresses obtained from unsteady Euler calculations. It was also shown that the effect of hot gas accumulation on the pressure surface is dependent on the flow coefficient. The pressure side temperature can be reduced by increasing the flow coefficient.

Other than the hot streak studies outlined above, a number of investigations on the effect of radial temperature distortions were undertaken by several researchers. Pappas [39] analyzed the experimental data taken at the MIT blowdown turbine facility with a circumferentially uniform, radially varying temperature profile. The heat flux into a transonic turbine rotor blade was directly measured. Several levels of radial distortion were tested including almost uniform inlet temperatures (RTDF factor less than 2%). It was shown that large levels of temperature distortion increased surface heat transfer, particularly on the pressure surface. Pappas used a 3-D steady state Euler Code to examine the rotor surface relative total temperature with the measured inlet profile as input. Streamline curvature calculations were also used to account the radial temperature variations. The 3-D calculation showed secondary flows in the rotor passage, resulting in the radial displacement of hot gas on the pressure from mid span toward both the hub and tip. However, the calculated surface relative total temperature did not completely account for the observed heat flux increase. Saxer [44, 46] calculated the flowfield of the same
transonic turbine with a 12% parabolic RTDF profile with a 3-D Euler unsteady solver using non-reflective boundary conditions.

Harasgama [28] investigated the influence of radially varying temperature profile on rotor aerodynamics and heat transfer, using a 3-D N-S code (Dawes [9]) and a 2-D boundary layer program (Stan5 [7]). Uniform as well as distorted inflow temperature conditions were input into the single blade row grid. A total pressure defect at the hub and tip was used at the rotor inlet to simulate the endwall boundary layer. Rotor tip clearance was set at 1% of span. The flow showed enhanced secondary flows in the rotor passage with the introduction of temperature distortion. At 50-60% chord, the mid-span hot fluid was swept to the tip region and carried over the tip clearance to the suction surface. Calculations with and without tip gap indicates that the tip leakage flow plays a major role in bringing midspan gas to the tip region.

The research work by previous investigators have greatly contributed to the understanding of how temperature distortion influences turbine durability. The gas segregation caused by circumferential distortion (Kerrebrock and Mikolajczak effect) leads to higher temperatures on the pressure side than the suction side. This effect can be reduced by increasing the flow coefficient (decreasing wheel speed). Secondary flows caused by radial temperature distortion, classical endwall vortices and tip leakage flow redistribute the midspan hot fluid toward the tip and hub walls.

Most of the studies in the temperature distortion problem were done analytically or numerically. The experimental work by Butler et al [5] at UTRC remains the only published data available on this subject. The poor agreement between calculations and data indicates a need in further understanding this phenomenon. Also does temperature distortion affect the local heat transfer coefficient? This would require the measurement of heat flux directly. Experimental data on high speed axial turbines, where durability concerns are the most severe, will undoubtedly broaden the scope of knowledge on this
1.3. Objective and Approach

The main objective of this thesis is to determine the effect of inflow nonuniformity on turbine rotor heat transfer. Of particular interest is the elucidation of turbine flow physics and scaling of distortion influence with turbine design variables. Specifically the influence of turbine inlet temperature distortion on rotor heat transfer will be examined. Low frequency temperature fluctuations (500 - 1000 Hz) shown in Fig. 1.3 may appear quasi-steady to the characteristic turbine vane passing frequency of 15-20 kHz. Therefore the current investigation focuses on steady temperature distortions only. The influence of momentum turbulence on rotor heat transfer in a rotating stage will also be determined.

The experimental work is carried out at MIT’s Blowdown Turbine Facility, using a highly loaded transonic turbine as test article. Temperature distortions typical of combustor exit flows are introduced at turbine inlet. Both radial and circumferential temperature distortions are examined independently to ascertain their influence. Turbine inlet turbulence is increased using a perforated grid, producing turbulence intensity and spectrum found in combustor exit flows. Blade surface heat transfer distribution is directly measured under various inlet conditions, using high frequency heat flux gauges. Aerodynamic data necessary for heat flux data reduction and analysis should also be taken.

Computational (CFD) tools are used to simulate turbine flowfield with results compared to experimental data. Influence of turbine design variables on turbine heat transfer is explored using the CFD tools on hand.
1.4. Thesis Outline

The remainder of the thesis is organized into the following chapters.

The experimental apparatus used for the current research program is discussed in Chapter 2. The basics of the blowdown turbine facility and the test turbine (ACE standing for Advanced Core Engine) are described. Particular emphasis is given to the aerodynamic and heat transfer instrumentation, as well as tunnel modifications necessary for distortion testing.

A brief description of the computational fluid dynamics (CFD) code used to study the flow phenomenon is offered in Chapter 3.

Chapter 4 describes the data reduction procedure and the assessment of data accuracy. Reduced turbine aerodynamic measurements, particularly the turbine inlet temperature, are thoroughly documented. The rotor surface heat flux data with inlet temperature nonuniformity are then presented, although detailed analysis are given in a subsequent chapter, Chapter 5.

A detailed study of the heat transfer data with radial and circumferential temperature distortion is given in Chapter 5. Flow physics that contribute to the measured heat flux changes are discussed. CFD simulations of the flowfield based on experimental conditions are used to interpret the heat transfer measurements. Heat transfer coefficients in the form of Nusselt number are calculated using reference gas temperatures predicted by different methods, and compared between uniform and distortion tests. Conclusions and recommendations are given with regard to blade heat load determination, as elucidated by the experimental results.

In light of the lessons learned from the experimental data, and considering the restraints imposed by current experimental setup, the effect of temperature distortion is
further investigated computationally, and presented in Chapter 6. Hot streaks with higher levels of temperature distortions and smaller length scale are simulated using the numerical tools on hand. New flow phenomenon that contribute to the hot streak redistribution and not previous reported are analyzed and presented. The influence on blade surface temperature (therefore heat load) are quantitatively assessed.

A final summary and conclusion are provided in Chapter 7, with the contribution of this thesis outlined. Recommendations for future work are also discussed.

Several appendices are included which supplement the main body of the thesis. Hot gas injectors' mass flow calibration are discussed in Appendix A. A simple error analysis of the heat flux are presented in Appendix B. Although not crucial to this thesis, some interesting topics that occurred during the course of this research are given in Appendices C and D.
Fig. 1.1: Trend of turbine inlet temperature with time. From [30]

Fig. 1.2: Current prediction methods under estimate rotor pressure surface heat transfer. Symbols represent different engine companies [19].
Fig. 1.3: Combustor exit dynamic temperature signals from an engine test. Average burner exit temperature 1200°K. From [16].

Fig. 1.4: Typical radial and circumferential temperature distribution for an annular burner at 2500°F. From [53].
Fig. 1.5: Measured CO2 concentration contours with and without introduction of temperature distortion. [5]
Fig. 1.6: Predicted mid-span rotor surface relative total temperature distribution using 2-D CFD codes. From [14].

Fig. 1.7: Predicted (3-D Euler and N-S Codes) and experimental time averaged surface temperature for rotor mid-span section [14].
Chapter 2.
Experimental Apparatus

This chapter describes the experimental apparatus used for the current research program. Modifications to the MIT blowdown turbine facility such as the temperature distortion generator and rake translators are particularly emphasized. Details of the aerodynamic and heat transfer instrumentation are thoroughly documented.

2.1. Blowdown Turbine Facility

The MIT blowdown turbine facility (BDT) is a transient wind tunnel capable of simulating flow conditions for most modern high pressure axial turbines. Details of the facility design and tunnel operation are given in references [17,24]. This section serves only to summarize the information helpful for subsequent sections and understanding of the circumstances under which the experimental data was taken.

The blowdown turbine facility was constructed as an alternative approach to continuous turbine testing. The test time in the BDT is less than a second. This time, however, is sufficiently long compared to the flow characteristic time in a turbine stage so that the turbine operates in a quasi-steady state. The test conditions are fully scaled and all the relevant nondimensional parameters important to the flow physics are simulated. Table 2-1 outlines the rig scaling for the current test article, the ACE stage.

The metal temperature is the same as room temperature because the test time is short compared to the thermal transient of the turbine blading. The ratio of specific heats, an important parameter for compressible flows, are matched by a gas mixture of Argon...
and Freon-12 (mass fraction of 51%). The gas-to-metal temperature ratio are kept the same therefore the heat lost to the blade is the same proportion of turbine enthalpy drop as in an engine environment. This sets the gas temperature of the main and cooling flows. The Reynolds number requirement determines the flow pressure. The Prandtl number is approximately the same as air at 1780°K.

Table 2-1: MIT Blowdown Turbine Scaling.

<table>
<thead>
<tr>
<th></th>
<th>Full Scale</th>
<th>MIT Blowdown</th>
</tr>
</thead>
<tbody>
<tr>
<td>Working Fluid</td>
<td>Air</td>
<td>Argon-Freon-12</td>
</tr>
<tr>
<td>Ratio of Specific Heats</td>
<td>1.28</td>
<td>1.28</td>
</tr>
<tr>
<td>Mean Metal Temperature, Tm</td>
<td>1118°K</td>
<td>295°K (Room Temp)</td>
</tr>
<tr>
<td>Gas/Wall Temperature Ratio, Tg/Tm</td>
<td>1.59</td>
<td>1.59</td>
</tr>
<tr>
<td>Inlet Total Temperature, Tg</td>
<td>1780°K</td>
<td>478°K</td>
</tr>
<tr>
<td>Cooling Air Temperature</td>
<td>790°K</td>
<td>212°K</td>
</tr>
<tr>
<td>True NGV Chord</td>
<td>8.0 cm</td>
<td>5.9 cm</td>
</tr>
<tr>
<td>Reynolds Number *</td>
<td>2.7 X 10^6</td>
<td>2.7 X 10^6</td>
</tr>
<tr>
<td>Inlet Total Pressure</td>
<td>19.6 atm</td>
<td>4.3 atm</td>
</tr>
<tr>
<td>Outlet Total Pressure</td>
<td>4.5 atm</td>
<td>1.0 atm</td>
</tr>
<tr>
<td>Prandtl Number</td>
<td>0.752</td>
<td>0.755</td>
</tr>
<tr>
<td>Rotor Speed, RPM</td>
<td>12,734</td>
<td>6,190</td>
</tr>
<tr>
<td>Eckert Number †</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>Mass Flow, kg/s</td>
<td>49.00</td>
<td>16.55</td>
</tr>
<tr>
<td>Power, watts</td>
<td>24.88 X 10^6</td>
<td>1.078 X 10^6</td>
</tr>
<tr>
<td>Test Time</td>
<td>Continuous</td>
<td>0.3 sec</td>
</tr>
</tbody>
</table>

* Based on NGV Chord and isentropic exit conditions
† (γ-1)M^2T/ΔT

Table 2-1 also shows the advantage of blowdown testing. Because of the lower gas temperature and the use of heavier working fluid, the turbine rotational speed is only half of that in continuous testing while still maintaining the corrected speed (or tip Mach
number). The slower rotor speed also means a lesser requirement on the bandwidth of high frequency instruments. The reduction in testing pressure and power generation makes the experimental facility safer and easier to manage. Another distinct advantage of the blowdown concept is that it readily provides heat transfer data. The large thermal inertia of the turbine hardware maintains an almost constant gas-to-metal temperature ratio during the test time without the application of airfoil cooling. The relatively benign environment due to the low gas and metal temperature facilitates the use of heat flux gauges as described in reference [18].

A schematic of the blowdown turbine facility is shown in Fig. 2.1. Major components shown from upstream to downstream are the supply tank, main fast acting valve, temperature distortion generator, upstream rake translator, test section and the dump tank. Cross-sectional views downstream of the main valve are given in Fig. 2.2 and Fig. 2.3. Temperature gradients are introduced by the radial temperature distortion generator (RTDFG) as well as hot gas injection. The upstream rake translator is located between the RTDFG and the test section. Another translator is installed downstream of the rotor, see Fig. 2.3. To maintain constant corrected rotor speed, the turbine power is absorbed by an eddy current brake, also shown in Fig. 2.3. The RTDFG, injectors and rake translators will be discussed in subsequent sections.

The flowpath is formed by concentric cylindrical walls. Upstream of the NGV is a flow contraction which simulates the aft portion of the combustion chamber. Bleeds upstream of the flow contraction intercepts the boundary layer upstream and restarts a new boundary layer into the test section. Exit flow of the rotor are again constrained by concentric cylindrical walls. The rotor exit flow conditions and turbine pressure ratio are determined by a downstream throttle plate, which slides along the tunnel center axis to adjust the choked exit flow area, see Fig. 2.3.

Before a typical test, the tunnel is evacuated. The supply tank, RTDFG and
injectors are heated to the required gas temperatures. Test gas mixture is then loaded into
the supply tank with the main valve closed. Translators, eddy current brake and A/D
systems are then set in standby mode. The rotor is spun-up in vacuum by the drive motor,
to above the desired test speed. After the drive motor is shut off, the rotor slows down
due to friction forces. When the rotor speed matches the preset value, timer circuitry is
activated which in turn opens the main valve, energizes the eddy current brake and starts
the data acquisition systems and translators. The main valve opens in about 30-50
milliseconds. The initial transient of the blowdown settles out in about 200 milliseconds.
The pressure difference between the supply tank and the dump tank sustains a test time of
about 300 milliseconds before the orifice downstream of the turbine stage unchokes.
During this time period, the turbine corrected speed and pressure ratio stay nearly constant
to better than 1%. The flow lasts approximately one second until the tunnel pressure
equilibrates. The data acquisition system continues to take data for ten minutes to monitor
tunnel conditions and to provide post test data for transducer calibration.

The test article being studied here is a 4-to-1 pressure ratio, transonic turbine
designed by Rolls-Royce, referred to as the ACE stage. Under design conditions, both the
NGV and rotor blade rows are choked. Some additional design parameters of this turbine
are given in Table 2-2. It should be noted that all dimensions given in this thesis are that of
the scaled model used in the BDT facility, with approximate scaling factor of 3/4.

Although the original turbine design included filming cooling on both the vane
and rotor blade and previous work included both cooled and uncooled rotor heat transfer
measurements [1], the current investigation utilized only the uncooled blading. Also a
rotor speed of 120% of the initial design was chosen as the operating point for this study
as suggested by Rolls-Royce. This leads to a rotor incidence angle reduction of 10° from
the original design value. There is also a change in the flow coefficient, i.e. the ratio of
axial velocity and wheel speed.
To facilitate data reduction of rotating instruments (translating rakes, rotor blade heat flux sensors), an angular coordinate is defined with the center-line of sector A as a zero reference and clockwise direction as positive, see Fig. 2.7. Under this definition, the three equally divided sectors, A, B and C would have annular coordinates of -60° to 60°, 60° to 180° and -180° to -60°.

Table 2-2: Design Parameters of ACE stage.

<table>
<thead>
<tr>
<th></th>
<th>NGV</th>
<th>Rotor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Airfoil</td>
<td>36</td>
<td>61</td>
</tr>
<tr>
<td>Mid-Span Axial Chord</td>
<td>3.4 cm</td>
<td>2.6 cm</td>
</tr>
<tr>
<td>Inlet Angle (From Axial)</td>
<td>0°</td>
<td>57°</td>
</tr>
<tr>
<td>Exit Angle</td>
<td>74°</td>
<td>-65°</td>
</tr>
<tr>
<td>Axial Gap</td>
<td></td>
<td>1.1 cm</td>
</tr>
<tr>
<td>Rotor Tip Diameter</td>
<td></td>
<td>55 cm</td>
</tr>
<tr>
<td>Turbine Load, $\Delta H/U^2$</td>
<td></td>
<td>2.3</td>
</tr>
<tr>
<td>Flow Coefficient, $C_x/U$</td>
<td></td>
<td>0.63</td>
</tr>
</tbody>
</table>

2.2. Radial Temperature Distortion Generator

The radial temperature distortion is introduced by a transient storage heat exchanger positioned in the flowpath upstream of the test section, see Fig. 2.2. Details of the design considerations can be found in [27]. The radial temperature distortion generator (RTDFG) consists of an annular stainless steel honeycomb matrix surrounded on its inner and outer diameters by jacketed walls, see Fig. 2.4. The honeycomb cell cross section and length are determined such that test gas exiting the heat exchanger is heated to the metal temperatures, and stays constant within the test time. Three mechanical struts supporting the inner annular assembly divides the flowfield into three equal sectors, defined as A, B and C as above. Oil or water can be fed into the jacketed walls to heat or cool the matrix.
boundary. Electrical heater cables are threaded through the honeycomb at the midspan center-line to produce a center peak temperature for radial temperature variation.

Three sets of electrical heaters are used, each controlling one sector of the flowfield. In principle, separate sectors can have different temperature distortions, and their effects on turbine heat transfer can be examined in a single test using the high frequency response heat flux gauges. In fact, heater cables in sector A are positioned in a discrete manner as shown in Fig. 2.4, aimed at producing both radial and circumferential temperature gradients. Tests indicated, however, that large circumferential temperature distortions could not be generated in this manner, due to the fact that the thermal conductivity in the circumferential direction is much higher than in the radial direction. This difference is a result of the fabrication method used in producing the honeycomb matrix. The matrix is rolled from alternating flat and corrugated stainless steel sheets, thus forming triangular flow passages as pictured in Fig. 2.5. The thermal conductivity variation resulted from the geometrical pattern and the poor thermal contact between the metal layers due to incomplete brazing. A heat conduction model (Appendix C) showed a 15 to 1 difference in conductivity in the two directions. Thus, circumferential temperature variations quickly turn into radial temperature distortions. While measures like increasing heating power and using cooling midway between heating positions enhanced the circumferential temperature gradient, it was concluded that the distortion level was too low. Eventually the circumferential temperature distortion had to be generated through hot gas injection, and will be discussed later.

The much higher thermal conductivity in circumferential direction is actually helpful in generating pure radial temperature distortions. The heat addition quickly forms a heat source along the center line and diffuses radially. The temperature profile in the honeycomb depends on the history of electrical power input. The modeling discussed in Appendix C provides a theoretical guide for adjusting the power levels to achieve various
temperature profiles. Thirty type J thermocouples were positioned at various locations inside the honeycomb to monitor the matrix temperature. For radial temperature distortion tests, neither heating nor cooling was applied to the jacketed walls of the RTDFG. For tests without radial temperature distortion, heating oil from the supply tank was fed into the jacketed walls to provide a nearly uniform honeycomb temperature, and electrical heating was used to add energy to the matrix to accelerate matrix temperature rise and shorten test preparation time.

This distortion generator was previously used in the experimental investigation reported in [27,39]. The unit was shown to be contaminated by large numbers of particles, which dislodged into the flow during each blowdown test. Consequently, the heat flux gauges used at that time were severely sand blasted and damaged. Ultrasonic excitation was used to shake the RTDFG while submerged in a plastic water tank. Large amounts of glass beads of 100 micron diameter was found on the bottom of the tank after the ultrasound excitation. This cleaning procedure was continued until no more particles were coming out of the matrix. Subsequent NGV tests, however, still showed unacceptable levels of "sand blasting" during a blowdown. Various options were considered to further decontaminate the RTDFG. The ultimate solution adopted was installing a particle filter directly downstream of the RTDFG. The filter used was a sintered stainless steel medium rated as blocking 20 micron diameter particles in a gas stream. This filter was supported by a downstream stainless steel perforated grid for mechanical safety. To minimize flow disturbance caused by the grid, the perforation pattern (1/8" round perforation with 3/16" center-center spacing, arranged in a hexagon pattern) was chosen as small as possible without adversely affecting the mechanical strength. The pressure drop across this filter/grid is very large, amounting to about 1/3 of the supply tank pressure, thus requiring an increased initial pressure in the supply tank.
2.3. Hot Gas Injection

Given the limitations of the RTDFG in generating circumferential temperature nonuniformity, an alternative scheme was devised as shown in Fig. 2.6. Essentially a bypass flow was created between the RTDFG entrance and particle filter exit. The pressure drop through the RTDFG and the filter provided the pressure gradient to sustain the bypass flow. An 15" long tube bundle heat exchanger, heated by externally wrapped heating tape, was used to increase the gas temperature. The high temperature gas was then injected into the main flow with 1.14" diameter injectors upstream of the upstream translating rakes, creating a hot spot like temperature distortion. The flow passages downstream of the heat exchanger was internally heated using coiled heating cables to avoid gas temperature drop. Perforated plate and honeycomb were installed for parallel and uniform flow at injector exit. The temperature distortion levels were limited by the chemical stability of Freon-12. At very high temperatures (800-1000°F), Freon-12 (dichlorodifluoromethane) begins to disintegrate, producing chlorine and fluorine, which oxidize metal surfaces and poses serious risk for tunnel components and instruments. Therefore the maximum gas temperature operable was in the range of 800°F (700°K). Immediately after the blowdown, room temperature Argon was fed into the hot gas injection system to further reduce the metal surface temperatures and reduce the risk of Freon disintegration.

A total of 4 injectors were installed inside sector A, see Fig. 2.7. The injector exit center line was positioned radially midway between the inner and outer annulus walls downstream of the boundary layer bleed, and circumferentially aligned with the center of NGV passages. The spacing between the injectors are 30°, with one hot spot per every 3 NGV’s. These injectors are numbered 1 through 4, see Fig. 2.6. The angular position of these injectors are approximately at -45°, -15°, 15° and 45°.
It is desirable to match the injector flow total pressure to that of the freestream. This means the flow resistance of the injection system should be tailored to that of the RTDFG and the filter. Injected flow temperatures are measured by stationary total temperature probed positioned at the injector exit center line. The mass flow is determined through the use of calibrated orifice plates installed inside the injector body. Pressure measurements upstream and downstream of the orifice, and exit gas temperature are used to deduce the mass flow. Details of the orifice calibration are described in Appendix A. Similarity among all 4 injectors is important to establish a periodic disturbance to the turbine rotor. This similarity is achieved during the assembly process and validated through the mass flow calibration. Type J thermocouples were positioned at various locations to monitor the temperatures of the heat exchanger and injector and guide the heating. Due to the lack of pressure transducers, orifice upstream pressures were only measured for injectors #2 and #3. These two measurements agreed with each other to within 0.5% of the averaged value, which is the accuracy limit of the pressure transducers used. An average pressure from injectors #2 and #3 was assumed in calculating the mass flows for injector #1 and #4.

2.4. Turbulence Grid

The objective here is to determine whether turbulence typical of engine combustor exit flows influences the heat load of turbine rotor blades operating in a multi-blade row environment. Turbulence measurements made by Moss and Oldfield [37] in a combustor were used as a reference in determining the appropriate turbulence intensity and length scale. Perforated plates are widely used as a turbulence source and this approach was taken here. Correlations between mesh properties (size, shape etc.) and turbulence characteristics by Roach [42] were used as a guide in the grid design.
rectangular bar grid was chosen as shown in Fig. 2.8. This is essentially a 1/4" plate with square holes of 0.75" by 0.75", yielding a porosity of 56%. This grid is supported on the lips of the boundary layer bleed, see Fig. 2.2. The turbulence intensity and spectrum were measured in a low speed wind tunnel and in the BDT facility. These measurements, presented together with the heat flux data later, confirmed the original design intent of generating an intensity and length scale comparable to those found in [37].

2.5. Circumferential Rake Translators

Knowledge of inlet temperature and pressure conditions is a basic requirement for studying the effects of distortion on turbine heat transfer. Circumferentially translating probes with multiple sensors in the radial direction allow a two-dimensional survey of the flowfield, and is the approach adopted here. Probe traversing downstream of the stage provides measurements necessary for deducing turbine aerodynamic performance. A downstream translator, with some adaptation, can also be used to measure the NGV exit flowfield with the rotor removed from the tunnel.

Given the extent of the inlet temperature distortion and the fact that different distortion types at different sectors are tested at the same time, it is desirable for the translator design to allow nearly 360° coverage. Other design considerations include, easy installation and removal of probes for calibration and repair, multiple probe capability for simultaneous temperature and pressure measurements, and constant traversing speed while taking data during a test. Probe position history during a test is required for data analysis. The final designs to meet these specifications vary some what between the upstream and downstream translators and will be discussed separately.
2.5.1. Upstream translator

Ideally the upstream measurement station should be as close to the NGV as possible and, in particular, downstream of the boundary bleeds. Many options were considered to package the driving mechanisms and signal cabling into the space available. This proved to be very difficult without major modifications to the test section. Eventually an alternate position was chosen as shown in Fig. 2.2. The translator was fitted into its own housing and flange mounted between the RTDFG and the main test section.

The final design is shown in the assembly drawing in Fig. 2.9. Rakes are directly mounted onto the rotating cantilevered flange, which is supported by large diameter slim-line bearings and driven by a bevel gear set with speed ratio of 6:1. The drive shaft located insider one hollow strut is connected to the externally mounted 5 kW electrical servo motor. An O-ring between the driving shaft and its inner stationary bearing seat provides the vacuum seal. Water cooling tubes are brazed onto the upstream flanges to terminate the heat flow from the heated RTDFG. This avoids any thermal loading of the moving parts inside the translator.

Electrical cables from the probes are bridged to and connected with the external casing mounted vacuum feed-through via a hollow strut, Fig. 2.9. Type K thermocouple and copper connection wires are used for the temperature and pressure probes. To prevent damage to the electrical cabling, a mechanical hard stop is installed which prohibits rake rotation of more than one turn. This hard stop also provides the reference position for initializing the translator.

Two probes are installed on the upstream translator, one 11 radial station total temperature probe and a three station total pressure probe, spaced 45° apart. The details of these probes are discussed in section 2.6.
2.5.2. **Downstream translator**

The challenge in designing the downstream translator was fitting everything in the space available, see Fig. 2.10. The original inner annulus wall downstream of the rotor blade row was replaced by a rotating drum, which is supported by large diameter slim line bearings and driven by an industrial chain drive. The driving shaft is positioned on the main test section flange and connects to the external motor via a 2:1 speed reducer gear box, see Fig. 2.1. The gear box reoriented the motor shaft to a position normal to tunnel centerline and increased the torque capability of the motor drive. The speed ratio of about 13.6 between the driving motor and the translator is determined by the gear box speed ratio, and the number of teeth of the sprockets (14 and 95).

The rake is designed to have a can-like base, with a 41 pin Bendix connector at the end of the can. All mechanical and electrical connections are secured inside the base. The probe then plugs into another hollow can (rake well) which is mounted on the translator. Electrical ribbon cables and vacuum reference tubes for pressure transducers from the mating connector at probe base are secured onto the translator drum and routed to the end of the test section, where the ribbons are looped in a hairpin fashion to accommodate the translator rotation, see Fig. 2.10 and Fig. 2.11. The final configuration shown in Fig. 2.11 is the result of many trial-and-error iterations. During a test, the rotating inner shroud pulls the ribbon to the insider diameter. After the test the translator must be returned to the starting position and to prepare for another test. This post test return was accomplished by sandwiching the ribbons with stainless shims. With the added stiffness of the shims, the inner shroud would be able to push the ribbon bundle and return to the starting position. As can be seen from Fig. 2.11, elaborate lining and tailoring was applied to ensure proper operation without buckling the ribbon bundle. This mechanism was fully contained and protected from the high speed discharge flow of the rotor, using small clearance seals between moving parts. A mechanical hard stop limits the rake motion to one revolution,
for the safety of the electrical wiring and for use as a position reference.

With the turbine rotor removed, this translator can be adapted to measure the NGV exit flow, as shown in Fig. 2.12. A cylinder whose diameter is the same as the NGV exit hub fills the gap left by the rotor and the rakes are moved forward closer to the NGVs.

Currently there are three 8-element rakes installed on the downstream translator, all at the same axial location and with an angular spacing of 20°. These include one low frequency total temperature probe and two total pressure probes with different frequency response characteristics. One hub static pressure tap in also available on the translating drum. The three rake probes are made such that they can be installed into any one of the three rake wells on the translator.

2.5.3. Translator motor drive

The accurate positioning of the translators were accomplished using feedback control as shown in Fig. 2.13. A Baldor DC servo motor (Model M4090B) was selected based on its torque-speed characteristics. An optical encoder is mounted onto the motor shaft. The encoder has a resolution of 4000 counts and a once-per-revolution index pulse. Copley Controls Corp. PWM servo amplifier provides the current power into the servo motor. The control loop is closed with a PC-bus compatible, single axis controller by Galil Motion Controls, Inc., Model DMC-400-10. The controller analyzes the motor position from the encoder and sends out a motor command signal to the amplifier, which in turn outputs an proportional current to the motor. Control parameters can be optimized to match the dynamics of each individual translator, using the diagnostics software supplied by the controller manufacturer.

The necessary input parameters for the controller are speed, acceleration and travel distance, all in terms of encoder counts. Once all these parameters are entered, the
motion can be started by software or by an external TTL trigger signal. The motor position can be polled from the controller at a frequency of 2 kHz. Fig. 2.14 shows a typical velocity profile and position trajectory of the upstream translator. Total travel of 320° was accomplished in 500 milliseconds, including 100 milliseconds of acceleration, 300 milliseconds coast, and 100 milliseconds deceleration time. The constant speed travel coincides with that of the scaled turbine test time. It should be noted that this travel schedule can easily be modified using the programs specifically written for this application.

The direction of travel during a run for both upstream and downstream translators are the same as turbine rotation, i.e. from sector C to A to B, see Fig. 2.6. Before each test, the translators are initialized to the desired starting position. The sequence of initialization is as follows: 1) reverse until against the hard stop; 2) forward to nearest motor once-per-rev index; 3) check distance between index and hard stop and compare with that previously recorded (This is a safety precaution); 4) move translator to starting position relative to the index position; and 5) set translator to stand-by mode, waiting for external trigger. All these operations are automated in the programs written for the translator operation.

2.6. Instrumentation and Data Acquisition Systems

The instrumentation in the blowdown turbine can be divided into three categories: a) basic tunnel condition monitoring sensors, b) tunnel pressure and temperature measurements essential for the data reduction and analysis for the current investigation, and c) rotor blade heat transfer gauges. The first category includes supply tank pressure and temperature, dump tank pressure, and various transducers for the eddy current brake operations. Details of this category can be found in references [17,24] and will not be discussed here. The last two categories will be discussed in the following sections.
2.6.1. Pressure and Temperature Measurements

Fig. 2.15 outlines major pressure and temperature measurement stations in the tunnel.

Inlet gas temperature is measured by an translator-installed 11 radial station type K thermocouple rake. Each sensor is shrouded inside a vented kiel head tube. The radial position of each sensor, given in Table 2-3, is designed such that the annulus area between them is the same from tip to hub. In addition to the translating temperature rake, four stationary thermocouple probes are positioned at the center of each injector exit. The reference junction for these thermocouple probes are contained in a stainless steel dewar filled with silicon oil and mounted external of the test section. Temperature signals are then amplified and recorded by a multiplexer-A/D system. Details of the signal conditioning, probe calibration and data reduction can be found in [52].

Table 2-3: Radial Positions of the Inlet Translating T/C sensors.

<table>
<thead>
<tr>
<th>Sensor Number</th>
<th>Radius (inches)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (Tip)</td>
<td>11.017</td>
</tr>
<tr>
<td>2</td>
<td>10.745</td>
</tr>
<tr>
<td>3</td>
<td>10.466</td>
</tr>
<tr>
<td>4</td>
<td>10.179</td>
</tr>
<tr>
<td>5</td>
<td>9.884</td>
</tr>
<tr>
<td>6</td>
<td>9.580</td>
</tr>
<tr>
<td>7</td>
<td>9.266</td>
</tr>
<tr>
<td>8</td>
<td>8.941</td>
</tr>
<tr>
<td>9</td>
<td>8.604</td>
</tr>
<tr>
<td>10</td>
<td>8.253</td>
</tr>
<tr>
<td>11 (Hub)</td>
<td>7.886</td>
</tr>
</tbody>
</table>

Two total pressure probes are used to measured inflow pressure field. A three
head pressure rake is mounted on the translator 45° away from the temperature probe, see Fig. 2.6. Another measurement is made using a stationary total pressure probe downstream of the boundary layer bleed in sector A. Table 2-4 details the sensors’ radial positions.

Table 2-4: Radial Positions of the Inlet Total Pressure Rake Sensors

<table>
<thead>
<tr>
<th>Sensor Number</th>
<th>Radius (inches)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 Translator</td>
<td>10.70</td>
</tr>
<tr>
<td>2 Translator</td>
<td>9.50</td>
</tr>
<tr>
<td>3 Translator</td>
<td>8.50</td>
</tr>
<tr>
<td>Stationery in Sector A</td>
<td>9.78</td>
</tr>
</tbody>
</table>

Pressure taps at the NGV exit measures the static pressures at the inner and outer walls between the NGV and rotor blade row. The angular position of these measurements are made at the center of the sector C. The position of these static taps, 0.040” in diameter, are shown in Fig. 2.16.

One total temperature and two total pressure probes are installed on the downstream translator, with 8 radial sensors with equal annular area spacing, summarized in Table 2-5. This measurement station is slightly more than 2 rotor axial chords downstream of the rotor trailing edge.
Table 2-5: Radial Positions of the Downstream Translator Rake Sensors

<table>
<thead>
<tr>
<th>Sensor</th>
<th>Radius (inches)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (Tip)</td>
<td>10.651</td>
</tr>
<tr>
<td>2</td>
<td>10.448</td>
</tr>
<tr>
<td>3</td>
<td>10.241</td>
</tr>
<tr>
<td>4</td>
<td>10.030</td>
</tr>
<tr>
<td>5</td>
<td>9.815</td>
</tr>
<tr>
<td>6 (NGV Hub Exit)</td>
<td>9.594</td>
</tr>
<tr>
<td>7</td>
<td>9.369</td>
</tr>
<tr>
<td>8 (Rotor Hub Exit)</td>
<td>9.137</td>
</tr>
</tbody>
</table>

Note: Outer and Inner wall radius
Rotor Exit: 10.85" (o) and 8.90" (i)
NGV Exit: 10.85" (o) and 9.42" (i)

The total temperature rake uses similar Type K thermocouple sensors as the upstream rake, with vented kiel head protecting the measurement junction. The reference junction is housed inside the cylindrical probe base, where a copper disk serves as a uniform temperature block. Transition from T/C to copper was made on solder pads glued onto the copper disk surface. The signal is then fed into the amplifier and recorded in the same manner as the inlet translating rake.

Two total pressure probes with high and low frequency response characteristics are installed on the translator to measure the rotor exit flow field. Kulite pressure transducers are used for these measurements. For the low frequency rake, the Kulites are installed inside the probe base and stainless steel tubes are used to connect the transducer to the Kiel head probes. To provide higher frequency response, a second total pressure probe is installed where Kulite transducers are flush mounted inside impact tubes at 8 radial locations identical to the temperature and low frequency pressure rakes. Both pressure rakes have back pressure reference tubes which are routed to outside the test section together with the electrical ribbon bundle.
Two additional static pressure measurements are made downstream of the rotor. A static tap on the downstream translator drum, located three rotor chords downstream of the rotor trailing edge, measures the hub static pressure. Static pressure on the outer wall are made through a flush mounted transducer, located at the center of sector C, one rotor chord downstream of the rotor trailing edge.

Upon removal of the rotor, the downstream translator can be refitted to measure the NGV exit flowfield. The downstream translating rakes are moved upstream to a position about 1.05" downstream of the NGV trailing edge. Because the rotor is replaced with an 18.84" diameter cylinder, the inner most two sensors of these rakes are not exposed to the NGV exit flowfield.

2.6.2. Rotor Blade Heat Flux Instrumentation

The rotor surface heat transfer rate is measured with MIT’s double-sided heat transfer gauges [17,18,24]. These gauges consist of thin film nickel resistance thermometers deposited onto a thin (25 μm) polyimide insulator known as Kapton, see Fig. 2.17. Surface heat flux is inferred from the temperature measurements of the "top" (exposed to the flow) and "bottom" (sandwiched between the Kapton and the blade) thermometers. At low frequencies, the temperature difference between the top and bottom sensors is directly proportional to the heat flux level. At high frequencies, only the top sensor temperature is necessary to deduce the ac heat transfer fluctuations. In practice a numerical procedure is used to calculate the heat flux from dc to a frequency relevant to the flow phenomenon of interest.

The Kapton sheets of gauges are bonded to, and completely cover, the anodized aluminum rotor blade. Three rotor blades were instrumented, with gauges at the nominal radial locations of tip, mid-span and hub. Fig. 2.18 illustrates the chordwise location of these sensors on the rotor blade profile. An composite drawing including sensors from all
three blades is given in Fig. 2.19. For each blade gauges are positioned at a constant radius from turbine's axis of rotation. The sensor's radial and chordwise is summarized in Table 2-6. The chordwise position is expressed in terms of surface distance from the leading edge. A "negative distance" is used for the suction surface sensors to distinguish from the pressure side for plotting convenience.

Table 2-6: Heat Flux Gauge Locations.

<table>
<thead>
<tr>
<th>Sensor #</th>
<th>Tip</th>
<th>Mid</th>
<th>Hub</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radius (inches)</td>
<td>10.534</td>
<td>10.050</td>
<td>9.605</td>
</tr>
<tr>
<td>% Span Rotor L.E.</td>
<td>79</td>
<td>46</td>
<td>16</td>
</tr>
<tr>
<td>Chordwise Position (Inch)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Sensor #</td>
<td>16</td>
<td>1.16</td>
<td>1.16</td>
</tr>
<tr>
<td>15</td>
<td>0.96</td>
<td>0.96</td>
<td>0.96</td>
</tr>
<tr>
<td>14</td>
<td>0.76</td>
<td>0.76</td>
<td>0.76</td>
</tr>
<tr>
<td>13</td>
<td>0.60</td>
<td>0.56</td>
<td>0.60</td>
</tr>
<tr>
<td>12</td>
<td>0.40</td>
<td>0.36</td>
<td>0.40</td>
</tr>
<tr>
<td>11</td>
<td>0.20</td>
<td>0.20</td>
<td>0.20</td>
</tr>
<tr>
<td>00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>01</td>
<td>-0.20</td>
<td>-0.20</td>
<td>-0.20</td>
</tr>
<tr>
<td>02</td>
<td>-0.40</td>
<td>-0.40</td>
<td>-0.36</td>
</tr>
<tr>
<td>03</td>
<td>-0.60</td>
<td>-0.60</td>
<td>-0.56</td>
</tr>
<tr>
<td>04</td>
<td>-0.76</td>
<td>-0.80</td>
<td>-0.76</td>
</tr>
<tr>
<td>05</td>
<td>-0.96</td>
<td>-1.00</td>
<td>-0.96</td>
</tr>
<tr>
<td>06</td>
<td>-1.16</td>
<td>-1.20</td>
<td>-1.16</td>
</tr>
<tr>
<td>07</td>
<td>-1.36</td>
<td>-1.40</td>
<td>-1.36</td>
</tr>
<tr>
<td>08</td>
<td>-1.56</td>
<td>-1.60</td>
<td>-1.56</td>
</tr>
<tr>
<td>09</td>
<td>-1.76</td>
<td>-1.80</td>
<td>-1.76</td>
</tr>
</tbody>
</table>

The details of the blade instrumentation, wiring and the necessary calibration of gauge thermometers are discussed in [22].
2.6.3. Data Acquisition Systems

Three A/D systems are used to record data for the current investigation.

1. MIT blowdown turbines 12 bit A/D system, with 8 multiplexers of 16 channels for each multiplexer. Data from the high frequency total pressure rake sensors and the top sensors of the heat flux gauges are sampled using this system. The low frequency pressure probes, bottom sensors of heat flux gauges are sampled using the multiplexers. This system is operated through a DEC μVAX II computer. The sampling frequencies for a typical run is the following:
   - 0-250 ms: 50 kHz
   - 250-550 ms: 200 kHz
   - 550-1200 ms: 50 kHz
   - 1200-10 min: 50 Hz

2. ADTEK Corp. Model AD-830 12 bit A/D system operated through a Dell 486 EISA IBM-PC. This systems provides additional high speed channels for the rotor blade heat flux instrumentation. This A/D is synchronized to the MIT BDT A/D with the same clocking and sampling rate.

3. Analogic model HSDAS-16 16 bit A/D card with AMUX-64-16 multiplexer. A total of 64 channels are multiplexed into a single high speed A/D converter. This system is operated through a Dell 486D/50 IBM-PC. and used for all the upstream and downstream temperature probes, temperature probes for injector flows and RTDs for the reference junctions. The sampling frequency of this system was set as the flowing
   - 0-1000 ms: 3.03 kHz
   - 1000 ms - 10 min: 1 Hz

4. In addition to the A/D systems mentioned above, the IBM-PC used for translator control also records the upstream and downstream translator positions in terms of encoder counts at a sampling frequency of 2 kHz. Programs were written to convert
these encoder positions into translator rake position files.
Fig. 2.1: Schematic of the MIT blowdown turbine facility.
Fig. 2.2: Cross-sectional view of the Blowdown Turbine facility internal flowpath, upstream of turbine stage.
Fig. 2.3: Cross-sectional view of the internal flowpath, downstream of turbine stage.
Fig. 2.4: Blowdown Turbine's Radial Temperature Distortion Generator
Fig. 2.5: Picture of the RTDFG honeycomb matrix.
Fig. 2.6: Schematic of the hot gas injection system.
Looking Downstream

Fig. 2.7: Turbine inlet flow geometry and injector locations. Mechanical struts divided flowfield into 3 equal sectors.
Fig. 2.8: Drawing of the turbulence grid. Three identical pieces can be installed.
Fig. 2.9: Upstream rake translator. The struts in the cutaway view are shown out of position.
Fig. 2.10: Downstream rake translator configured for rotor exit flow measurement.
Fig. 2.11: Downstream translator electrical ribbon loop containment. The ribbon indicated consists of three electrical ribbons sandwiched between 0.003" stainless shims for stiffness.
Fig. 2.12: Downstream rake translator configured for NGV exit flow survey.
Fig. 2.13: Functional diagram of the translator position control.

Fig. 2.14: Upstream translator speed and position trajectory for a typical blowdown test. Angular position defined in Fig. 2.7.
Fig. 2.15: Aero instrumentation layout in blowdown turbine. Nomenclature in () indicating probe designation.
Fig. 2.16: NGV exit static tap positions. Drawing not to scale.

Fig. 2.17: Multi-layer heat flux gauge. Drawing not to scale.
Fig. 2.18: Heat flux gauge locations around the rotor blade.

Fig. 2.19: Rotor blade heat flux gauge sensor location and status.
Chapter 3.
Numerical Procedure

To better understand the flow physics and help interpret the experimental observations, a computational fluid dynamics code was used to numerically simulate turbine flow field. The code used is an unsteady three dimensional, multiple blade row Euler solver written by Saxer [46]. The flow geometry is discretized into an unstructured H-H type grid of hexahedral cells, upon which the fully nonlinear Euler equation set is solved using the Ni-Lax-Wendroff time marching scheme. The computational grid is shown in Fig. 3.1 and Fig. 3.2. The numerical algorithm is implemented in conservative form, meaning that mass, momentum and energy are conserved throughout the flow field. Smoothing is applied for shock capturing and to prevent saw-tooth oscillations in the solution. Non-reflective boundary conditions are used at inlet, exit and blade row interface. For steady state calculations, mass, momentum and energy across the stator rotor interface are conserved using an circumferential stream-thrust flux averaging technique, therefore retaining the radial variations while smearing out flow gradient in the pitchwise direction. This treatment is sometimes called the mixing plane method. For unsteady (time accurate) calculations, variations in both radial and circumferential directions are accounted for using the non-reflecting boundary conditions.

The equation set is solved in nondimensional form. The flow geometry is nondimensionalized by the axial chord of the vane at the hub ($L_{ref} = 33.8$ mm). Flow variables are normalized by reference quantities of density $\rho_{ref}$, speed $c_{ref}$. Introducing these quantities gives the following non-dimensional variables, with ' denoting dimensional variables,
Length: \[ x, y, z = \frac{x', y', z'}{L_{\text{ref}}}. \]

Velocity: \[ u, v, w = \frac{u', v', w'}{c_{\text{ref}}}. \]

Density: \[ \rho = \frac{\rho'}{\rho_{\text{ref}}}. \]

Pressure: \[ p = \frac{p'}{\rho_{\text{ref}} c_{\text{ref}}^2}. \]

Enthalpy: \[ h = \frac{h'}{c_{\text{ref}}^2}. \]

Angular Speed: \[ \Omega = \frac{\Omega'}{c_{\text{ref}} / L_{\text{ref}}}. \]

Eq 3.1

For uniform inlet conditions, \( \rho_{\text{ref}} \) and \( c_{\text{ref}} \) are the NGV inlet stagnation density and stagnation speed of sound respectively. Using this definition, the inlet stagnation pressure and stagnation enthalpy are \( 1/\gamma \) and \( 1/(\gamma - 1) \), with \( \gamma \) being the ratio of specific heats. When the flowfield is not uniform, the stagnation density and stagnation speed of sound at the hub on the periodic boundary are used as reference.

Certain input conditions are required for the non-reflective boundary conditions. At the stage inlet: These are radial and tangential flow angles, stagnation enthalpy and stagnation pressure (or entropy). At the stage exit: static pressure at the hub is required and a simple radial equilibrium is assumed. The rotor speed of rotation is needed in the proper nondimensional form. For all the calculations done here, the stage inlet tangential flow angle (from axial) is set at zero while radial flow angle is set to be parallel to the inner and outer wall slope at hub and tip and interpolated at other radial positions. The inlet temperature field, rotor speed and rotor exit static pressures are obtained from experimental measurements.

The computational domain consists of 3 NGV passages and 5 rotor passages,
shown in Fig. 3.2. This corresponds to 36 NGVs and 60 rotor blades, comparing to 36 NGVs and 61 rotor blades in the experimental hardware. The rotor therefore is scaled by a factor of 61/60. The NGV grid block has $56 \times 36 \times 21$ (axial, circumferential and radial) points for every stator passage, and $56 \times 22 \times 21$ points for every rotor passage. This amounts to 256,368 grid points in total. Rotor tip clearance is not modeled here.

Steady state calculations require about 5000 iterations to reach a converged solution, with rms. residual of $\sim 10^{-6}$. Unsteady calculations need about 7-10 periods (time for the rotor grid to move through 3 NGV passages). Pressures from one grid point on each side of the stator-rotor interface are recorded and examined to check for periodicity. Generally the maximum non-periodicity of these pressures is of $10^{-5}$ for a converged unsteady solution.
Fig. 3.1: Computational grid for the ACE stage, axial radial plane.
Fig. 3.2: Computational grid in the pitchwise direction.
Chapter 4.
Data Reduction and Experimental Results

A series of tests were carried out to ascertain the influence of inlet temperature distortion on turbine blade heat transfer. Data obtained include time resolved rotor heat flux as well as tunnel pressure and temperature. In particular, the NGV exit flowfield was measured using the downstream translator probes. This chapter describes the data reduction procedure and presents the reduced heat transfer data. The measured effects of temperature nonuniformity on turbine heat transfer are discussed without elaborate explanation. Further detailed analysis will be given in a subsequent chapter.

4.1. Test Matrix and Tunnel Conditions

Because the aerodynamic rakes are traversed circumferentially and heat flux gauges produce time resolved data, the test section was configured into 120° annular sectors divided by struts, with different inlet temperature conditions in each sector. For instance, high temperature gas was injected into sector A, thus creating hot-spot-like temperature distortion, while the other two sectors (B and C) have uniform inlet conditions. Therefore, data with and without temperature distortion are taken at the same time. Table 4-1 summarizes the full stage tests presented in this thesis. Also included in Table 4-1 are the mass-averaged turbine inlet temperature, derived from inlet translating rake measurement and injector massflow and temperature data. The nature of the distortion in each sector is noted as uniform, RTDF (radial temperature distortion factor)
or $\theta$TDF (circumferential temperature distortion factor). Conditions in sector C are similar to that of sector B in all tests, although detailed temperature data are not available due to incomplete coverage by the upstream translator rakes. Effects of radial temperature distortion or turbulence can be assessed by comparing heat flux data in sector B between tests (T169$^1$ vs. T174 or T175 vs. T174). $\theta$TDF effects can be analyzed between tests T171 and T174, or between sector A and sector B in test T171 alone. It will be shown in later sections that the second approach is preferred because of the better data accuracy.

Table 4-1: Full Stage Test Matrix

<table>
<thead>
<tr>
<th>Test ID</th>
<th>Inlet $T_T$ from station 1, see Fig 2.15</th>
<th>Sector A</th>
<th>Sector B</th>
<th>Tu Grid</th>
</tr>
</thead>
<tbody>
<tr>
<td>T169</td>
<td>*</td>
<td>RTDF, $T_{T,\text{mass,avg}} = 498^\circ\text{K}$, $T_{T,\text{max}} = 576^\circ\text{K}$</td>
<td>no</td>
<td></td>
</tr>
<tr>
<td>T171</td>
<td>$\theta$TDF $T_{T,\text{mass,avg}} = 504^\circ\text{K}$, $T_{T,\text{hot spot}} = 657^\circ\text{K}$</td>
<td>Uniform $T_T = 488^\circ\text{K}$</td>
<td>no</td>
<td></td>
</tr>
<tr>
<td>T173</td>
<td>Uniform $T_T = 488^\circ\text{K}$</td>
<td>Uniform $T_T = 488^\circ\text{K}$</td>
<td>no</td>
<td></td>
</tr>
<tr>
<td>T174</td>
<td>$\theta$TDF $T_{T,\text{mass,avg}} = 497^\circ\text{K}$, $T_{T,\text{hot spot}} = 561^\circ\text{K}$</td>
<td>Uniform $T_T = 488^\circ\text{K}$</td>
<td>no</td>
<td></td>
</tr>
<tr>
<td>T175</td>
<td>$\theta$TDF $T_{T,\text{mass,avg}} = 505^\circ\text{K}$, $T_{T,\text{hot spot}} = 657^\circ\text{K}$</td>
<td>Uniform $T_T = 489^\circ\text{K}$</td>
<td>YES</td>
<td></td>
</tr>
</tbody>
</table>

* With single proof of concept injector installed

Upon completion of stage testing, the rotor was removed from the test section. The downstream translator was refitted to measure the NGV exit flowfield. Test conditions listed in Table 4-1 were repeated and NGV exit pressure and temperature were measured. The NGV exit Mach number, calculated from NGV exit tip and hub static pressure measurements, were matched to that of full stage tests by adjusting the

---

1 $T###$ designates a particular test number
downstream throttle position. These NGV only tests are summarized in Table 4-2. Turbine inlet temperatures are repeatable to within 5°C from stage tests, acceptable for heat transfer data analysis. Temperature distortion levels are repeatable to 10%. Two tests with and without the turbulence grid (T186 and T187) were carried out to measure the turbulence spectrum, using hot wire probes installed upstream of the NGV leading edge. The inlet gas temperature was set as room temperatures to increase the hot wire overheat ratio and improve measurement accuracy [8].

### Table 4-2: NGV Only Test Matrix.

<table>
<thead>
<tr>
<th>Test ID</th>
<th>Inlet Conditions</th>
<th>NGV Exit Measurement</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>T181</td>
<td>A: 0TDF, ( T_{T, \text{mass,avg}} = 499^0 \text{K} ) ( T_{T, \text{hot,spot}} = 642^0 \text{K} )</td>
<td>Traverse over 3rd Spot</td>
<td></td>
</tr>
<tr>
<td>T183</td>
<td>B: Uniform, ( T_T = 487^0 \text{K} )</td>
<td>Traverse in Sector B</td>
<td></td>
</tr>
<tr>
<td>T185</td>
<td>B: RTDF, ( T_{T, \text{mass,avg}} = 500^0 \text{K} ) ( T_{T, \text{max}} = 568^0 \text{K} )</td>
<td>Traverse in Sector B</td>
<td></td>
</tr>
<tr>
<td>T186</td>
<td>B: Uniform, ( T_T = 300^0 \text{K} )</td>
<td>Traverse in Sector B</td>
<td>HotWire w/o Grid</td>
</tr>
<tr>
<td>T187</td>
<td>B: Uniform, ( T_T = 300^0 \text{K} )</td>
<td>Traverse in Sector B</td>
<td>HotWire with Grid</td>
</tr>
</tbody>
</table>

4.1.1. **Temperature measurements**

Details of the temperature data reduction and uncertainty analysis were discussed by Sujudi [52]. It was shown that time-averaged temperature measurements are accurate to ±1-2°C in sectors with uniform or radially varying temperatures. The frequency response of the upstream and downstream translating temperature rakes, however, was limited due to the high thermal inertia of the thermocouple junctions. With the inlet temperature rake traversing at nominal speed (thus providing 240° coverage), the temperature profile of the injector flow was not sufficiently resolved to allow accurate
determination of the hot spot and averaged temperature data in sector A. Fortunately, the mass averaged total temperature could be evaluated using injector mass flow and temperature measurements by stationary thermocouple probed.

Uniform inlet temperature is introduced by heating the RTDFG's jacketed inner and outer annulus walls with oil. The inlet temperature field in sector B is shown as contours in Fig. 4.1, for test T173. The flow field is uniform except for a 15°K radial temperature variation, equal to a 2% RTDF. This variation, existed for all uniform temperature tests, is caused by the heat loss through RTDFG outer casing. This heat loss resulted in lower matrix temperatures at the tip.

The injector flow properties for circumferential distortion test (T171) are summarized in Table 4-3. The temperature pattern measured by the inlet translating rake is shown in Fig. 4.2. The low frequency response of the temperature sensor results in the "tails" apparently trailing each hot spot. This is an instrumentation induced artifact. Injector mass flow was measured using a calibrated orifice plate located inside each injector. The total mass flow in sector A is determined from inlet total pressure and temperature and by assuming a choked NGV throat area of 22.4 in\(^2\) (the Rolls Royce design value.) The hot spot temperature variation among the injectors is of the order of ±10°K, or 5% of the spot-to-freestream temperature difference. Because injector mass flow scales with gas temperature, the enthalpy flux variation among the injectors is smaller, less than ±2%. The slow response time of the downstream temperature rake prohibited accurate temperature measurement across a complete 120° sector downstream of the NGV. Therefore, an angular extent equal to three NGV passages (one hot spot pitch) is surveyed more accurately by reducing translator speed. The NGV exit temperature distribution is shown in a contour plot, see Fig. 4.3. A maximum to minimum temperature ratio of 1.1 was measured. The shape is very similar to that shown in Fig. 1.4, although the magnitude variation is less. The same data is replotted in Fig. 4.4 against
pitchwise position at three representative spanwise locations. As can be seen, the hot spot has mixed out considerably from the injector exit to downstream of NGV. The temperature at the edge of the spot is still at the freestream level (before hot gas injection), indicating that the individual spots have not merged into each other. The question is where did the mixing occur? To answer this question, a high temperature hot wire (used as temperature probe, or cold wire at low overheat ratio) was traversed radially at an axial location 0.81" upstream of the NGV leading edge. The measured hot spot center line radial temperature variation is shown in Fig. 4.5. Also plotted are the NGV exit peak temperatures measured at six radial positions by the traversing rake. The agreement is within the hot wire measurement uncertainties of 5-10°K. This figure suggests that the radial temperature distribution is not altered by passage through the NGV.

Table 4-3: Freestream and Injector flow properties, T171.

<table>
<thead>
<tr>
<th>Injector</th>
<th>Mass Flow (kg/s)</th>
<th>$T_T$, °K</th>
</tr>
</thead>
<tbody>
<tr>
<td>Freestream</td>
<td>4.23*</td>
<td>488</td>
</tr>
<tr>
<td>Injector #1</td>
<td>0.0947</td>
<td>666</td>
</tr>
<tr>
<td>Injector #2</td>
<td>0.0908</td>
<td>654</td>
</tr>
<tr>
<td>Injector #3</td>
<td>0.0937</td>
<td>647</td>
</tr>
<tr>
<td>Injector #4</td>
<td>0.0920</td>
<td>660</td>
</tr>
</tbody>
</table>

* For sector A only, excluding mass flow from injectors

During initial injector development, the jet mixing was estimated using correlation by Abramovich [3]. For flow conditions in Table 4-3, the jet should retain its unmixed core temperature along the center line for 13 jet diameters downstream of the injector exit. The blowdown turbine injectors are located 14 jet diameters upstream of the NGV. The mixing measured in the facility is considerably greater than the calculations suggest. The cause may be the high turbulence levels generated by the particle filter and grid, which is
immediately upstream of the injectors. The NGV exit measurement indicates a circumferential temperature distortion of 10% OTDF.

Radial temperature distortion is introduced through electrical heating at midspan of the honeycomb. The temperature field of this radial distortion is shown in Fig. 4.6 for sector B in test T169. Temperature variation is predominately in radial direction, with a small circumferential variation at midspan. The circumferentially averaged radial variation, shown in Fig. 4.7, has an RTDF factor of 28%. Also shown is the radial temperature distribution for the uniform test, T173. This measurement plane is quite far from the NGV leading edge, about 4 annulus heights upstream. To quantify the temperature distortion at the rotor inlet, an NGV only test (T185) was performed with similar inlet conditions, see Fig. 4.8. The measured NGV exit temperature variation, plotted in Fig. 4.9, is smaller in magnitude than that measured upstream, as much as 50%. This may be caused by mixing due to high levels of turbulence generated by the particle filter and grid. The NGV exit flow survey for uniform inlet temperature is also shown for comparison. Heat transfer into the NGV airfoil and tunnel end walls resulted in gas enthalpy drop across the NGV. The lower temperatures near the hub and tip for uniform inflows are also caused by heat loss to tunnel walls. It is interesting that the radial profile at NGV exit is skewed toward the hub, similar to that used by Harasgama [28]. Based on NGV exit flow measurements, the radial distortion factor is about 12%, typical of modern combustor exit conditions.

4.1.2. Pressure measurements

Heat transfer is a function of flow Reynolds number, which is proportional to turbine inlet total pressure. For the tests reported in Table 4-1, the averaged inlet total pressure is repeated within 0.5%, also see Table 4-5. The degree of uniformity of inlet total pressure is also of interest. Measurements taken by the inlet translating rake showed total pressure variations less than the measurement uncertainty, typically at 0.5%. For the
low NGV inlet Mach number of 0.07, flow dynamic head is only about 0.3% of the total pressure. Therefore, the inlet total pressure measurement serves to verify that there is no abnormal flow conditions such as separation or blockage. NGV and rotor exit static pressure measurements showed no variation of rotor inlet and exit Mach number changes due to temperature nonuniformity, see Table 4-4.

Table 4-4: NGV and Rotor Exit Mach Numbers.

<table>
<thead>
<tr>
<th>Test ID</th>
<th>NGV exit</th>
<th>Rotor exit</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>tip</td>
<td>hub</td>
</tr>
<tr>
<td>169</td>
<td>1.11</td>
<td>1.32</td>
</tr>
<tr>
<td>171</td>
<td>1.11</td>
<td>1.32</td>
</tr>
<tr>
<td>173</td>
<td>1.12</td>
<td>1.32</td>
</tr>
<tr>
<td>174</td>
<td>1.12</td>
<td>1.32</td>
</tr>
<tr>
<td>175</td>
<td>1.12</td>
<td>1.33</td>
</tr>
</tbody>
</table>

4.1.3. Turbulence measurement.

The turbulence grid was designed to produce a turbulence intensity and spectrum distribution similar to that reported by Moss and Oldfield [37]. Grid performance was evaluated in a low speed wind tunnel [21] prior to installation in the blowdown facility. The turbulence level in the blowdown turbine facility, with and without the turbulence grid, was determined through hot wire measurements at a location 0.81" upstream of the NGV leading edge [8]. The measured turbulence intensities are 2-3% without the grid and 7-8% with the grid in the rig. Facility turbulence of 0.5% was previous measured before the installation of RTDFG and particle filter. The spectrum is compared to that by Moss and Oldfield in Fig. 4.10. Overall, the turbulence created met the original design intent.
4.1.4. **Summary of overall turbine operating conditions**

During the 300 milliseconds test time, the rotor corrected speed and pressure ratio stay constant to within 1%. The heat flux data are extracted from a time window equivalent to 10 rotor revolutions between 440-530 milliseconds. Main turbine operating conditions are given in Table 4-5. The rotor inlet relative total temperature is based on midspan NGV exit conditions using a simple velocity triangle calculation.
Table 4-5: Stage Test Tunnel Conditions.

<table>
<thead>
<tr>
<th>Test ID</th>
<th>T169</th>
<th>T171</th>
<th>T173</th>
<th>T174</th>
<th>T175</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sector</td>
<td>B</td>
<td>A</td>
<td>B</td>
<td>A</td>
<td>B</td>
</tr>
<tr>
<td>RTDF or 0TDF</td>
<td>12%&lt;sup&gt;R&lt;/sup&gt;</td>
<td>10%&lt;sup&gt;θ&lt;/sup&gt;</td>
<td>0%</td>
<td>0%</td>
<td>-</td>
</tr>
<tr>
<td>Tu. Grid</td>
<td>n</td>
<td>n</td>
<td>n</td>
<td>n</td>
<td>n</td>
</tr>
<tr>
<td>Rotor speed (rev/s)</td>
<td>120.2</td>
<td>124.0</td>
<td>124.0</td>
<td>122.7</td>
<td>122.3</td>
</tr>
<tr>
<td>Mean Inlet T&lt;sub&gt;r&lt;/sub&gt;, °K</td>
<td>498</td>
<td>504</td>
<td>488</td>
<td>488</td>
<td>497</td>
</tr>
<tr>
<td>Rotor Rel. T&lt;sub&gt;r,rel&lt;/sub&gt;, °K*</td>
<td>431</td>
<td>435</td>
<td>421</td>
<td>421</td>
<td>429</td>
</tr>
<tr>
<td>Corrected Speed % Design</td>
<td>114</td>
<td>117</td>
<td>119</td>
<td>119</td>
<td>117</td>
</tr>
<tr>
<td>T&lt;sub&gt;r,rel&lt;/sub&gt;/T&lt;sub&gt;metal&lt;/sub&gt;</td>
<td>1.38</td>
<td>1.37</td>
<td>1.33</td>
<td>1.33</td>
<td>1.36</td>
</tr>
<tr>
<td>Inlet P&lt;sub&gt;n&lt;/sub&gt;, atm</td>
<td>3.69</td>
<td>3.67</td>
<td>3.66</td>
<td>3.65</td>
<td>3.68</td>
</tr>
<tr>
<td>Pressure Ratio π</td>
<td>3.97</td>
<td>4.00</td>
<td>4.00</td>
<td>4.00</td>
<td>4.03</td>
</tr>
</tbody>
</table>

* Calculated from averaged inlet gas temperature and assuming an averaged NGV exit flow Mach number of 1.22 and flow angle of 74°.

### 4.2. Heat Flux Data Reduction and Uncertainty Estimate

#### 4.2.1. Data reduction

Rotor surface heat flux is measured using the double-sided heat flux gauges shown in Fig. 2.17. Surface heat flux is inferred from temperature measurements on both sides of the Kapton insulator. At low frequencies (dc-20 Hz), the top and bottom temperature difference is directly proportional to the heat flux level, the Kapton insulator acts like a thermal shunt. At high frequencies (above 20 kHz), the bottom sensor is sufficiently shielded by the Kapton insulator and only the top sensor temperature is necessary to deduce the ac heat transfer fluctuations. In practice a numerical procedure
was used to calculate heat flux from DC to a frequency relevant to the flow phenomenon of interest.

The first step in reducing heat flux data is to determine the gauge sensor temperature history. For this test series the temperature is determined through a procedure called post-run calibration. This calibration technique will be described in some detail later because it directly affects heat flux data uncertainty. The top and bottom temperature data are then low-pass filtered, to a cut-off frequency of 20 kHz for top sensors and 2 kHz for bottom sensors. The turbine's NGV passing frequency is 4.3 kHz. Raw heat flux data is obtained using the top and bottom sensor temperatures and knowledge of the thermal physical properties of the Kapton insulator, specifically the $k/d$ ratio and \( \sqrt{\rho c_k} \). At this point, time unsteady heat flux data is obtained during the 300 millisecond test time when turbine's corrected conditions are constant. From this period, ten rotor revolutions of time varying heat flux data are then ensemble averaged to produce a segment of data that represents the periodic surface heat flux while the blade rotates 360°. Since each individual rotor blade passes through three different 120° sectors with different inlet conditions on each rotation, the ensemble averaged data can be further divided into sectors for analysis. For all the tests presented here, a time averaged heat flux value was obtained by averaging slightly less than one third of the ensemble averaged data, corresponding to a centered 90° portion of each 120° annular sector. Only 90° was used to avoid the influence of the wakes from the mechanical struts.

4.2.2. Heat flux data uncertainty

Most of the uncertainty in this data stems from uncertainty in assessing the accuracy of gauge sensor temperatures. For a typical blowdown test, all gauge sensors start at the isothermal blade temperature (room temperature), which is measured by the resistance temperature devices (RTD’s) mounted under the platforms of the instrumented
blades. During the initial blowdown transient, the gauge temperature increases rapidly, due to the compressional heating associated with tunnel startup pressure rise. After this transient settles out, the mean gauge sensor temperature continues to increase due to heat addition to the rotor blade. Eventually the blade reaches a maximum temperature of about 100°C. As the tunnel gas cools down after the test (due to expansion and heat loss to the walls) and the rotor rotation stops, the blade surface temperature decreases because of the large thermal inertia of the rotor blades and disk assembly. At three minutes after the run starts, the blade surface temperature is sufficiently uniform to within 0.1°C, as indicated by the well-behaved bottom gauge sensors. The temperature of the test gas, measured by the temperature probe downstream of the rotor, is close to the rotor blade surface temperature. Thus heat transfer into (or out of) the rotor blade due to natural convection is minimal at this time. From this we can conclude that the top and bottom temperatures are the same during the post-run temperature decay. This fact that all heat flux gauge sensors (on a single blade) start at the same temperature before the blowdown starts (the first 0-30 milliseconds of recorded data) and track each other during the temperature decay period provides a unique method of verifying the gauge temperature accuracy and most importantly, provides a calibrating technique.

All bottom sensors performed very well throughout this test series. The calibration constants obtained in a constant temperature bath calibration are used to convert raw A/D voltage into temperature. The measurements from these bottom sensors support the foregoing argument that the blade surface temperature is uniform to within 0.1°C before the test starts and during the post run temperature decay period, thus becoming valuable references in the gauge temperature calibration. The top sensors, however, suffer aging during and between tests. The pre-rig assembly temperature bath calibration eventually becomes useless. Fortunately, a new set of calibration constants for these top sensors can be obtained for each test by assuming that their temperatures are
equal to the blade surface temperature during the post-run temperature decay. Since all the bottom sensors agree well with each other, a single bottom sensor was chosen as an indicator of the blade metal temperature, and all the top and bottom sensors on each blade are forced to match this reference sensor. This post-run calibration procedure is validated if the indicated sensor temperatures before the start of the blowdown agree with each other. The top and bottom sensor temperature mismatch then becomes a measure of accuracy of the gauge temperature calibration.

Among the current test series, T174 and T175 had the best gauge performance. After the post-run calibrations, the temperatures indicated by the heat flux gauges all agree to better than 0.5°C before the blowdown starts. It is important to point out that for heat flux measurements, the accuracy requirement for the top and bottom sensors are the relative accuracy between them (i.e. the accuracy of the temperature difference between the sensors). A top to bottom temperature difference of 5°C during the test with a 0.5°C mismatch before the test represents a 10% heat transfer uncertainty. The absolute temperature of the top sensor (blade wall temperature) is important only in deriving the heat transfer coefficient (since the difference between the gas and wall temperature is important here). Typically this difference is 100°C to 130°C. Thus for a 5% heat transfer coefficient accuracy, the absolute wall temperature need only be accurate to within 5°C or so. A simple uncertainty analysis was performed for test T174 on the accuracy of the heat flux measurements, taking into account of the uncertainties in thermal properties of the Kapton insulator, specifically, the k/d ratio, and gas temperature uncertainties. This analysis is offered in Appendix B. For Test T174, all the heat flux sensor measurements are accurate to 5-10%.

For the reminder of the test series, the top sensors suffered abnormal electrical resistance changes during the initial blowdown transient. These resistance changes show up during data reduction as temperature variations. After the post-run calibration, the
temperature mismatch before the test is large enough that were the heat flux uncertainty to be based on this measurement alone, it could be as much as 200%! However, since the sensor erosion occurs mostly before the heat flux data is taken (i.e. during the tunnel startup transient) and a post-run calibration procedure is used, the accuracy of the sensor temperature during the run is much less affected. If there are no abnormal resistance changes after the test, the heat flux data reduced from a post-run temperature calibration can be, in principle, as good as that for test T174. However, the assessment of data uncertainty must be done using other techniques.

The fact that different sectors of the annulus are operated with different inlet conditions provides an opportunity to compare data between tests. In particular, the circumferential temperature distortion tests were carried out with uniform inlet temperatures in two sectors (B and C). In other words, heat flux data with uniform inlet temperature were taken during most of the tests. The data quality of these tests can then be assessed through proper nondimensional comparison with test T174 data.

It is important to point out that, if these abnormal resistance changes occur as point events before or after the run, the error in the top sensor temperature would appear as a direct DC shift. This will result in an error in the average heat transfer level. The unsteady fluctuations should still be accurate however. For tests in which uniform and distorted inlet conditions coexist in different sectors, a direct comparison can be made with baseline uniformity, provided the mean heat transfer level is not too far off. The effect of circumferential distortion was addressed in this way, as will be discussed later.

To compare heat flux data taken during different tests, heat flux is nondimensionalized into a Nusselt number using the following formula.

\[
Nu = \frac{\dot{Q}L}{k_{\text{wall}}(T_{\text{gas}} - T_{\text{wall}})}
\]

Eq 4.1
where $\dot{Q}$ is heat transfer rate per unit area. The rotor axial chord at mid span, $L$ is used as a length reference ($L = 25.725$ cm). $k_{wall}$ is the gas conductivity evaluated at the blade surface temperature, and $T_{wall}$ is that indicated by the top sensor of the heat flux gauge. Although $T_{gas}$ normally represents the adiabatic temperature of the boundary layer, a better known quantity, rotor relative stagnation temperature is used here. This temperature, often referred to as the inlet bulk temperature $T_{T,rel}$, was calculated from the mass averaged inlet total temperature measured by the inlet traversing rake, taking into account the frame of reference transformation from absolute to rotor relative. This frame transformation uses the NGV exit flow Mach number (calculated from the average of static pressure measurements at the tip and hub NGV endwalls) and a flow angle of $74^\circ$ (derived from a streamline curvature calculation). This rotor relative total temperature is given in Table 4-5.

As mentioned before, test T174 has the lowest heat flux data uncertainty. Heat transfer to the same rotor was measured at mid-span during a previous test program using different instrumentation which exhibited high stability and therefore good data accuracy, test T52. The test conditions of T174 and T52 are very close to each other, particularly the corrected rotor speed and inlet total pressure (thus Reynolds number), Table 4-6. The stage pressure ratio differed somewhat, reflecting a difference in downstream throttle setting, as can be seen from the rotor exit absolute Mach number. At the current rotor speed, both the NGV and rotor blade row are choked. Therefore the difference in throttle setting should not affect the rotor flowfield upstream of the blade throat.
Table 4-6: Test Conditions for Tests T174 and T52.

<table>
<thead>
<tr>
<th></th>
<th>T174</th>
<th>T52</th>
</tr>
</thead>
<tbody>
<tr>
<td>Corrected rotor speed, %</td>
<td>118</td>
<td>120</td>
</tr>
<tr>
<td>Stage pressure ratio, π</td>
<td>4.00</td>
<td>4.27</td>
</tr>
<tr>
<td>Rotor exit flow Mach number</td>
<td>0.52</td>
<td>0.60</td>
</tr>
<tr>
<td>Inlet total pressure, atm</td>
<td>3.65</td>
<td>3.56</td>
</tr>
<tr>
<td>Inlet total temperature, °K</td>
<td>488</td>
<td>463</td>
</tr>
</tbody>
</table>

A comparison of the rotor heat transfer at midspan is shown in Fig. 4.11. The estimated uncertainty bar of each measurement is also shown. The agreement between the tests is excellent. The higher heat transfer coefficient for test T174 on the suction surface close to the trailing edge is caused by the lower rotor exit Mach number, due to a different throttle setting. This is in agreement with measurements by Ashworth [4] on the same blade geometry.

Three tests were conducted at different levels of circumferential distortion (tests T171, T173 and T174). Test T171 has the highest injector exit gas temperature. In test T173, the injector gas temperature was set to that of the freestream flow (no circumferential temperature distortion). Test T174 had a temperature distortion level intermediate between those of tests T171 and T173. The inlet temperature field in the B sector is uniform in the three tests so that heat flux with uniform inlet conditions can be extracted from all three tests. Tests T171 and T173 have unacceptable heat flux data uncertainties based on temperature calibration estimates alone, but comparison of their data with that of test T174 can elucidate the data quality of these two tests.

Fig. 4.12 through Fig. 4.14 show the comparison among the tests of heat flux distribution along the blade surface at three spanwise locations. The uncertainty levels for Test T174 are indicated. Although many data points from test T171 fall outside the
uncertainty levels of test T174, a majority of the T171 measurements do agree quite well with that of T174. Test T173 agrees with test T174 better than test T171. The suction surface heat flux is more repeatable than that on the pressure surface for these three tests. The relatively large discrepancy among a few of the pressure side data points of the tip section clearly indicates that those sensors suffered erosion during the test time. The good agreement between tests for the majority of the sensors suggests a data correction, such as forcing the test T171 data points to the levels seen on tests T173 and T174. There are only a few data points like this, however, so that such corrections were not applied here to avoid biasing the data. The data repeatability through these three tests is within 5-10%, which is quite good for heat transfer measurements. Unfortunately, comparisons like these can not be made for the radial temperature distortion test T169, since no uniform sector is available for direct comparison. However, it does not seem to be overstating to conclude that a majority of the sensors should have data quality comparable to these three tests.

As was pointed out earlier, gauge erosion only contributes to the data uncertainty of the time-averaged heat transfer levels. The accuracy of the ac heat flux is not affected. Stage inlet conditions of test T171 was repeated in another test, T178. The heat flux at two locations on the blade pressure surface at mid span are plotted in Fig. 4.15 and Fig. 4.16. The excellent ac repeatability is obvious between the two tests, particularly near the leading edge. At locations near the trailing edge, however, unsteadiness related to boundary layer instability contributes to the randomness of the ac fluctuations. The bias in the averaged heat transfer is clearly seen in Fig. 4.15, caused by aging of the top sensor.

Since both uniform and distorted inflow conditions exist in test T171, the influence of circumferential distortion may be assessed using data from a single test, improving data accuracy. However, do minor geometrical differences exist between the two sectors which would resulting in differing heat flux? It is known that small changes in the NGV throat area lead to relatively large unsteady heat flux change[2]. Also the flow
injectors and the supporting strut may introduce perturbations into the flow. The magnitude of this effect has been estimated as follows. Test T173 has uniform inlet temperature around the entire annulus. Thus the agreement in time averaged heat flux among the three sectors in this test is a measure of the geometric uniformity as it influences heat transfer. Fig. 4.17 through Fig. 4.19 indicate that the time averaged heat flux around the annulus is very similar at all spanwise positions. It is thus concluded that geometrical differences in this rig contributed little to the time averaged heat transfer, when averaged across nine NGV passages, as has been done with this data.

In summary, the relative heat flux measurement accuracy in this study is 5-10% between tests. The effect of circumferential distortion can be analyzed by comparing data from sectors A and B of test T171, avoiding the dc bias associated with sensor erosion. The influence of radial temperature distortion and influence of grid turbulence must be compared across different tests.

4.3. Heat Transfer Measurements

The time-averaged heat transfer coefficient (Nusselt number) using averaged rotor inlet total temperature (see Table 4-5) is presented in this section. This section will show the measured heat transfer changes caused by temperature distortion, with detailed analysis postponed to the next chapter.

4.3.1. Surface heat flux distribution with uniform inlet Temperature

This data set extends the coverage of the blade surface heat transfer measurement for the ACE turbine to other than mid-span, in particular no hub data had been previously available. It should be pointed out that while the mid and tip sections of the rotor instrumentation do approximately follow the axisymmetric streamlines (i.e. as predicted by
streamline curvature codes), the hub section represents a cylindrical surface of constant radius. At the rotor leading edge, the hub heat flux gauges are located at 16% span from the inner annulus wall. At rotor trailing edge, these gauges are 36% from the hub wall.

Knowledge of the spatial distribution of rotor surface heat transfer is crucial to turbine cooling design. Fig. 4.20 shows the chordwise Nusselt number distribution for the nominal tip, midspan and hub sections of the blade. The chordwise and spanwise variations are very large, as much as 40%. The heat load on the pressure surface is higher than on the suction surface for all three blade sections. The radial variation has opposite trends on the pressure and suction surfaces. On the pressure side, the highest heat transfer rate is at the hub. On the suction surface, however, the hub section has the lowest heat load. Fig. 4.20 suggests that, if radial variations in heat transfer coefficient is not accounted for, heat transfer errors of 40% can result.

4.3.2. Influence of grid generated turbulence

It is well known that grid generated turbulence can increase heat transfer rates on cascade surfaces, as shown for instance by Doorly [10] and Ashworth [4]. The transition point is brought forward toward the leading edge on both the pressure and suction surfaces, therefore increasing heat transfer into regions with otherwise laminar boundary layer. For a turbine stage however, the rotor is subject to a different complex unsteady flow condition, including NGV wakes and shocks. Ashworth et al [4] simulated the NGV-rotor interaction on stationary rotor blades, with upstream NGV shocks and wakes generated by an array of bars rotated on an off center disk. The rotor blade geometry is the same as that of the ACE rotor blade. Tests with and without grid turbulence were also undertaken. Rotor frame turbulence intensities of 0.8% and 4% was tested. They found that the unsteady wake and shock system increased the rotor surface heat transfer by 50-100% over front portions of the airfoil, comparing to data without shock and wake
passing. The introduction of grid turbulence (increasing Tu from 0.8% to 4% in rotor frame), together with passing bars, increased the heat transfer rate by 20% at front portions of the pressure surface, compared to tests with passing bars alone. This finding implies that change of turbulence from 0.8% to 4% would increase rotor heat transfer by 20% in an stage environment, since the influence of the NGV was modeled through passing bars. To the knowledge of the author, there are no published data from a rotating rig, addressing how turbulence affects turbine rotor heat load. The objective here was therefore to introduce a velocity turbulence level comparable to that of common combustor exit flows, and measure the effect on the rotor heat transfer. Including test T52, midspan heat transfer data was measured with turbulence intensities of 0.5%, 3% and 8%. There is a small enthalpy drop through the cold grid (relative to the flow). This drop is on the order of 2°K, not significant for heat transfer data analysis since the gas to wall temperature difference is between 100 and 150°K. The time-averaged heat transfer coefficient with and without turbulence is shown in Fig. 4.21 through Fig. 4.23. Comparison is made between tests T175 and T174 with uniform inlet conditions (sector B). As is readily seen in the steady state data, there is no change in the rotor heat transfer. A few data points on the pressure side at midspan have very high levels of uncertainty. The time-resolved heat flux at one midspan gauge location is also compared in Fig. 4.24. There is no recognizable influence on the heat transfer history either.

This data does not necessarily contradict the findings for cascade heat transfer by Ashworth et al [4]. For the current investigation, the turbulence level was varied between 0.5 and 8% in front of the NGV. Since the mean flow velocity has increased by as much as ten fold from inlet to exit, the turbulence level at the NGV exit may be much less than the 4% turbulence tested by Ashworth et al. Nevertheless, if the turbulence introduced here is indeed comparable to that of typical combustor exit flow (i.e. if the data by Moss and Oldfield are representative of modern combustor exit conditions), then the influence of
velocity turbulence on rotor heat transfer is negligible in gas turbine engines.

4.3.3. **Influence of circumferential temperature distortion on rotor heat transfer**

Hot gas was injected into an annular sector equal to one third of the flowpath. The ensemble averaged heat flux for one rotor revolution with circumferential distortion and with uniform inlet temperatures are compared for two midspan gauge locations in Fig. 4.25 and Fig. 4.26. The heat flux is plotted against rotor blade angular position. The portion centered between -60° and 60° corresponds to the sector with circumferential distortion. The four hot gas streams clearly result in the increased heat transfer level. The four peaks related to hot gas injection are more pronounced on the pressure surface, as compared to the suction surface. Note that heat transfer between the hot spots are at the same level as without distortion (test T173), suggesting that the spots do not mix into each other. The high frequency fluctuations associated with NGV-rotor interaction are largest close to the rotor leading edge, making the temperature distortion effects less visible. This is especially true at the crown of suction surface.

As discussed before, time-averaged heat transfer comparison between uniform and circumferential distortions can be made using data from a single test T171. Since the mass flow through the injectors is well calibrated, the inlet mass averaged total temperature into one sector is well known. Because the free stream total temperature is the same around the annulus before injection, the averaged total temperature in the distorted sector is higher by 3% due to hot gas injection. This difference in averaged enthalpy is accounted for in calculating the Nusselt number. Fig. 4.27 through Fig. 4.29 compares the heat transfer coefficient with circumferential distortions to that with uniform inlet flow. At mid-span, there is a small increase in heat transfer for both the pressure and suction surface. The pressure surface overheating is slightly higher, about 10%. At the
hub, there is no measurable change in the heat transfer level on the suction surface, while the pressure surface heat transfer increases roughly the same amount as at midspan. There is little change in heat transfer on the tip section.

4.3.4. Effect of radial temperature distortion on rotor heat transfer

A circumferentially uniform, radial temperature distortion was introduced around 360° of the tunnel annulus. Thus, comparison with uniform inlet conditions must be made against another test. Sector B of test T174 is chosen as a reference since it represents the lowest uncertainty heat transfer data. For the radial temperature distortion test, T169, heat transfer in the same sector (sector B) is averaged for comparison.

For test T169, the mass averaged total temperature is used to calculate a bulk $T_{r,ref}$ for the reference gas temperature in Nusselt number calculation. The bulk gas to wall temperature ratio is 3% higher in the test with distortion (T169) than the undistorted reference (T174).

The time-averaged data are shown in Fig. 4.30 through Fig. 4.32. The heat transfer level at mid span is approximately 20% higher than that with uniform inflow, reflecting the higher than average temperature at mid span. The time averaged temperature at the tip is also higher than in the uniform case. The bulk Nusselt number is again higher around the blade surface at the tip. However, the increase is larger on the pressure surface than on the suction surface, by as much as 20% of the mean level. At the hub, the averaged total temperature at 16% span is actually lower than that of the uniform case. The pressure surface heat transfer increased by about 15% while the suction surface heat transfer decreased by 25% to 30%. Flow mechanisms that contribute to the observed heat transfer variation at the tip and hub will be address in Chapter 5.
4.4. Summary

A set of detailed measurements have been made on the influence of inlet temperature distortion on turbine rotor heat transfer. Detailed aero as well and heat transfer measurements were taken and data uncertainties were assessed and validated. The experimental data can briefly summarize as follows.

1. Circumferential temperature distortions up to 10% TDF with length scale of one NGV pitch were created at stage inlet. This level of circumferential distortion had little influence on the rotor blade suction heat transfer but increased pressure side heat transfer coefficient up to 10%. (Here heat transfer coefficient means Nusselt number based on inlet average bulk averaged temperature)

2. Circumferentially uniform, radially varying temperature distortion of up to 12% changes the local rotor blade heat transfer coefficient by ±20%.

3. The measured rotor heat transfer was not affected by varying the NGV inlet turbulence intensity from 0.5% to 8%.

This data set is thus far the only high speed turbine data addressing the issue of combustor generated temperature distortion influence on turbine heat transfer. The measured effects are large enough to warrant inclusion into turbine designs. More detailed analysis of this data will be given in the next chapter, where CFD tools are used to simulate the flow field. Flow mechanisms that influence rotor heat load will be addressed.
Fig. 4.1: Upstream total temperature field for uniform conditions.

Fig. 4.2: Total temperature contours measured by the inlet translating rake at the injector exit plane.
Fig. 4.3: NGV exit hot spot total temperature contours (domain of 3 NGV passages).
Fig. 4.4: NGV exit temperature distribution downstream of a hot spot measured at three spanwise locations.
Fig. 4.5: Radial variation of hot spot center line temperature measured upstream and downstream of NGV.

Fig. 4.6: Inlet total temperature contours for the radial distortion test, T169 sector B.
Fig. 4.7: Radial temperature variation for uniform and radial distortion tests.

Fig. 4.8: Radial temperature profile for full stage and NGV only tests.
Fig. 4.9: NGV exit radial temperature distribution for uniform and radial distortion tests.

Fig. 4.10: Turbulence power spectral density generated by grid. Measurement with dual hot wire probe.
Fig. 4.11: Comparison of current data and previous measurements of rotor Midspan heat transfer.

Fig. 4.12: Midspan heat transfer with uniform inlet temperature - Data Repeatability.
Fig. 4.13: Tip heat transfer with uniform inlet temperature - Data repeatability.

Fig. 4.14: Hub heat transfer with uniform inlet temperature - Data repeatability.
Fig. 4.15: Repeatability of Time Resolved Heat Transfer. Midspan Sensor #11.
Fig. 4.16: Repeatability of Time Resolved Heat Transfer. Midspan Sensor #14.
Fig. 4.17: Midspan heat transfer at three 120° sectors - Tunnel uniformity in $\theta$.

Fig. 4.18: Tip heat transfer at three 120° annular sectors - Tunnel uniformity in $\theta$. 
Fig. 4.19: Hub heat transfer at three 120° annular sectors. Tunnel uniformity in $\theta$.

Fig. 4.20: Rotor surface heat load vs. chord and span. Uniform inlet temperature. Data taken from test T173 sector B.
Fig. 4.21: Effect of grid turbulence on rotor heat transfer - Midspan.

Fig. 4.22: Effect of grid turbulence on rotor heat transfer - Tip section.
Fig. 4.23: Effects of grid turbulence on rotor heat transfer - Hub section.
Fig. 4.24: Pressure side comparison of grid turbulence influence at midspan gauge #14.
Fig. 4.25: Suction Side heat Flux Comparison of $\theta$TDF Heating Influence.
Fig. 4.26: Pressure side heat flux comparison of θTDF heating influence.
Fig. 4.27: Influence of OTDF on rotor heat transfer - Midspan.

Fig. 4.28: Influence of OTDF on rotor heat transfer - Tip section.
Fig. 4.29: Influence of θTDF on rotor heat transfer - Hub section.

Fig. 4.30: Influence of RTDF on rotor heat transfer - Midspan.
Fig. 4.31: Influence of RTDF on rotor heat transfer - Tip section.

Fig. 4.32: Influence of RTDF on rotor heat transfer - Hub section.
Chapter 5.
Heat Transfer Data Analysis and Comparison with CFD Results

5.1. Introductory Remarks

Before analyzing in detail the rotor heat transfer data taken under different temperature conditions, it is worthwhile to examine how temperature nonuniformity affects the flowfield of an axial flow turbine. For the transonic stage studied here, the NGV passage is choked and the flowfield is steady, except a portion of the suction surface downstream of the throat. With temperature distortions at constant stagnation pressure, the NGV flow pattern is the same as that without distortion, as predicted by the Munk and Prim substitution principle. The temperature variation is simply convected along streamlines.

The most obvious change in the rotor flowfield is the relative inlet flow angle. This is illustrated in Fig. 5.1. The NGV exit flow speed is proportional to the square root of the local gas temperature (constant Mach number). Upon subtraction of rotor wheel speed, both the relative flow angle and flow velocity magnitude change. As a result, hot fluid has a relative slip velocity directed to the pressure surface. Cold fluid, on the other hand, migrates toward the suction surface. With circumferential temperature distortions, the flow angle changes with time. Hot gas migrates toward the pressure surface and spreads out along chordwise directions, causing higher time averaged gas temperature on the pressure surface. This mechanism was first analyzed by Kerrebrock and Mikolajczak to
explain rotor wake transport in the downstream stator passage of axial compressors. For axisymmetric radial temperature distortions, rotor inlet flow angle changes along the span. The relative motion of gases with different temperatures causes a secondary flow inside the rotor passage, where hot gas at midspan moves toward the pressure surface and migrates to the hub and tip endwalls. Another explanation of this thermally driven secondary flow is through examination of the streamwise vorticity. The velocity gradient caused by radial temperature variation downstream of the NGV introduces an vorticity vector that is normal to the absolute flow direction (no secondary flows in the NGV). This vorticity has a streamwise component in the rotor relative frame and therefore causing secondary flows. Strength of the secondary flow is further modified according to the theories by Hawthorne[29]. This flow redistribution mechanism affects blade local heat load because the blade surface gas temperature deviates from that upstream of the leading edge at the same span, therefore heat transfer rate.

The change in rotor incidence angle changes the rotor surface pressure distribution and blade loading. This pressure distribution change may modify the state of the boundary layer, and therefore the heat transfer coefficient as well. Cascade heat transfer measurements of the same rotor geometry were made in a transient wind tunnel by Ashworth [4]. Rotor incidence angles at design and off-design values were tested. The flow angle was varied from design+5°, design and design-10°. This measurement, included here as Fig. 5.2, showed that the suction surface heat transfer coefficient is sensitive to incidence changes while pressure surface heat transfer is not (within the incidence angle ranges tested). The -10° incidence angle change corresponds to an increase of rotor speed from 100% to 120%. Data from MIT's blowdown facility for 100% and 120% rotor speeds [1] showed large heat flux variations at the front portion of the suction surface, Fig. 5.3. There is a small (5-10%) change in pressure surface heat transfer. Unfortunately, the instrumentation for the distortion tests yielded poor coverage on the crown of the
rotor blade. For the temperature distortion levels (both the radial and temperature distortions) in this study, the rotor inlet flow angle variation is on the order of $3^\circ$.

The influence of temperature distortion on heat transfer can be divided into two categories. First, the flow redistribution changes the local gas temperature, thus the driving temperature across the boundary layer. Secondly, the local heat transfer coefficient may be altered due to changes in the flowfield. The relative importance of these two mechanisms can be assessed by nondimensionalizing the heat flux into Nusselt number, using local gas temperatures as reference temperatures

$$Nu = \frac{\dot{Q}L}{k_{wall} (T_{ref} - T_{wall})}$$

If there is no change in the heat transfer coefficient, Nusselt numbers for uniform and distorted flows would be the same. Any difference in the Nusselt numbers requires examination of the flow to determine whether it is due to uncertainties in gas temperature prediction or to the viscous boundary layer behavior.

The calculation of gas temperature at the blade surface can be done with streamline curvature methods (relatively simple) or CFD codes (relatively complex and time-consuming depending on the type of code). The use of streamline curvature results addresses the basic question: Can a 2-D section of the 3-D airfoil be designed using the inlet gas temperature at each spanwise station to estimate the heat load and determine the cooling design. If the surface temperature deviates from the leading edge value, the effect of this local gas temperature variation on heat flux can be quantified using the following heat transfer influence coefficient

$$Q_T = \frac{T_{r,local} - T_{wall}}{T_{r,le,avg} - T_{wall}}$$

Eq 5.1
For the ACE stage tested here, the ratio between relative gas temperature to metal
temperature is about 1.35, see Table 4-5. This definition of influence coefficient is
motivated by the fact that heat transfer is proportional to the temperature difference across
the boundary layer, rather than the absolute temperature levels. The value of $Q_t$ is an
indication of the fractional increase in heat transfer caused by gas temperature variation
from leading edge values, therefore an indication of error in heat load if the streamline
curvature approach is used.

Because the radial temperature profile was directly measured at the NGV exit, the
rotor relative gas temperatures at each of the tip, midspan and hub blade sections were
estimated from the measured temperature profile through simple velocity triangle
calculations. This approach, similar to a streamline curvature procedure, will be referred to
as the 2D streamline method. A 3-D multiple blade row Euler solver, written by Saxer
[46] will be used to ascertain the effect of fluid redistribution in the rotor passage. A brief
description of the code was given in Chapter 3. Several authors examined the difference of
steady and unsteady calculations [46,50]. The choice of steady or unsteady method has
significant implications in design because faster steady codes are preferred in an iterative
process. However, the flow physics often dictates the choice depending on whether
unsteadiness is an important part of the flow phenomenon. The focus of this study is to
interpret heat flux data using CFD as a tool. Since both time-averaged and time-resolved
flow quantities are of interest, only unsteady calculations are presented. Averaged flow
variables (such as gas temperature) are obtained by time-averaging the instantaneous
values from the unsteady solution.
5.2. Rotor Surface Heat Transfer with Radial Temperature Distortion

5.2.1. Time-averaged heat flux data

Heat transfer variations along the span clearly indicate that the radial temperature gradient modifies the heat load, as was shown in Chapter 4 in fig. 4.30 through fig. 4.32. Therefore the first step in correlating heat flux between radial distortion and uniform tests is to account for this radial temperature variation. The NGV exit temperature measurement shown in fig. 4.9, is used to calculate the relative total temperature for each of the nominally tip, midspan and hub sections of the instrumented rotor blade, through simple velocity triangle calculations, i.e. the 2D streamline approach. For comparison, a rotor inlet "bulk" temperature, calculated from the average of NGV exit temperature, is used to calculate a baseline Nusselt number.

Data from radial distortion (T169) and uniform flows (T174) are presented in Fig. 5.4 through Fig. 5.6. At midspan, Fig. 5.4, heat transfer coefficients based on 2D streamline approach are nearly the same for uniform and distorted flows, meaning that the raw heat flux increase here is attributed to gas temperature changes. The 2D streamline method does a good job on the tip section suction surface in collapsing Nusselt numbers onto that of uniform flows, Fig. 5.5. On the pressure surface of the tip section, however, Nusselt number with radial distortion is consistently higher than the uniform flow case. At the hub pressure surface, Nusselt number is nearly 50% higher, Fig. 5.6. The chordwise averaged Nusselt numbers from three spanwise positions are shown for both the pressure and suction surfaces in Fig. 5.7 and Fig. 5.8. The influence of radial temperature distortion varies in the radial direction, with the strongest effect at the hub. This radial dependence suggests that designs based on streamline curvature method can lead to large errors in surface heat load estimates at the pressure surface, especially near the hub.
The basic assumption of streamline curvature approach is that total temperatures remain constant along 2-D axisymmetric stream surfaces. Fig. 5.6 suggests that the gas temperature may change down the chord, with higher temperatures on the pressure surface and lower temperatures on the suction surface. Thus may be caused by secondary flows resulting from temperature distortion. This fluid migration process associated with secondary flows is an inviscid phenomenon and can be modeled using an Euler solver. A parabolic temperature profile fit from NGV outlet data, Fig. 5.9, is assumed at the NGV grid inlet. The radial temperature distribution upstream and downstream of the NGV row should be the same, as predicted by the Munk and Prim principle. This is essentially a validation of the numerical procedure. The flowfield with uniform inlet conditions is also calculated for comparison. A snapshot of the static pressure field in the blade-to-blade plane at midspan is shown in Fig. 5.10. There are many common flow features between the two calculations, for instance the shock wave rotor blade interaction. Shocks impinge on the rotor surface and reflect many times during one blade passing period. It was previously established that this shock system is responsible for the large magnitude ac heat transfer fluctuations [2].

The circumferentially averaged rotor inlet flow angle and relative total pressure variation in the spanwise direction are presented in Fig. 5.11 and Fig. 5.12. The secondary flows initiated by the temperature distortion is visible from the temperature contours in the radial-tangential plane of the rotor passage at different axial stations, shown in Fig. 5.13. The relative motion of the midspan high temperature fluid toward the pressure surface and subsequent radial spreading is indicated, especially at the hub where temperature gradient is the highest. This causes the divergent streamlines on the pressure surface and convergent streamlines on the suction surface with distortion, Fig. 5.14. The streamlines for uniform inlet temperatures are evenly spaced along span, see Fig. 5.15. The secondary flow mechanism redistributed hot and cold temperature fluid on the blade surfaces, as
shown in Fig. 5.16. The blade surface temperature contours for the uniform case are also shown in Fig. 5.17 for comparison. With distortion, pressure surface temperature near the trailing edge is higher than values at leading edge of same span, resulting in higher heat transfer rates.

The surface temperatures at heat flux gauge locations were interpolated from the CFD calculation and used to normalize the measured heat flux into Nusselt numbers. The results are plotted in Fig. 5.18 through Fig. 5.20 for midspan, tip and hub sections. The CFD derived reference gas temperatures do not noticeably improve the collapse of the distorted data toward the uniform inflow data as compared to using the 2D streamline approach at the tip. The hub section pressure surface Nusselt numbers based on CFD calculations are nearly identical with and without distortion, implying the increased heat flux was caused by increased surface temperatures. The heat flux difference on the suction surface, however, remains unexplained.

Rather than just attributing the unexplained heat transfer to changes in the viscous boundary layer, let us examine if the assumptions in the Euler approach contribute to errors in calculating surface temperatures. It was noted in the literature review of Chapter 1 that, the endwall secondary flows associated with endwall boundary layer as well as tip leakage flow all contribute to the fluid redistribution in the rotor passage. But none of these effects were modeled by the 3-D Euler code herein. Harasgama [28] analyzed the flowfield of a transonic 4-1 pressure ratio turbine with tip clearance of 1% span, Using an N-S code developed by Dawes [9]. The calculation showed that tip leakage flow draws the midspan high temperature gas into the tip region and carries it over the tip gap to the suction side, see Fig. 5.21. This would cause higher pressure surface temperatures. The rotor as tested in the MIT blowdown turbine facility has tip clearance values between 1 to 2%. Therefore the tip leakage flow is expected to be stronger than that reported in [28]. To test if this phenomenon is applicable to the ACE rotor, the tip pressure surface
heat flux is normalized using midspan gas temperature as reference and compared to uniform test data. Fig. 5.22 shows excellent agreement between the uniform and distortion data. Modeling of the tip clearance flow against the experimental data using a validated Navier-Stokes code would undoubtedly be very helpful in confirming this observation.

Another flow feature not modeled by an Euler code is the endwall secondary flows. The vorticity contained in the inner and outer annulus boundary layers forms a horse-shoe vortex pair upon interaction with the rotor leading edge. Rotor turning transforms this vortex pair into the so-called "passage vortex" as it travels downstream within the rotor passage. The secondary flows caused by this passage vortex system tend to transport midspan fluid into endwall regions on the pressure surface and squeeze flows into the midspan region on the suction surface, similar to the effect of the secondary flows caused by radial temperature distortion. This mechanism was demonstrated in CFD analyses [54,55,14] and visible in Butler's CO$_2$ concentration data with uniform inlet conditions, Fig. 1.5a. The flow visualization result by Ashworth[4] in a linear cascade using ACE rotor geometry also showed strong secondary flows, particularly on the suction surface where the endwall flow influence is evident up to 30% span, Fig. 5.23. This endwall vortex system adds to the hot/cold fluid redistribution caused by temperature distortion, and therefore qualitatively explains the heat transfer coefficient difference shown in Fig. 5.20. A Navier-Stokes code validated for endwall simulation can be used to examine whether this additional effect will "quantitatively" explain the higher heat transfer rates on the hub pressure surface and lower heat flux on the suction surface. There are two aspects worthy of further studies: 1) Are the effects of radial temperature distortion generated secondary flows and endwall secondary flows additive? 2) How do the two effects interact with each other. For example, does the endwall vortex position change as a result of the introduction of temperature distortion?
5.2.2. **Time-resolved heat transfer**

The ensemble-averaged heat transfer data based on the NGV passing frequency for tests with and without radial distortion are presented in Fig. 5.24 through Fig. 5.26. The wave-forms are essentially the same shape independent of temperature distortion at virtually all gauge locations. This implies that there are no major changes in the flow structure, only the mean level is changed. The magnitude of unsteady fluctuations (with the same wave-form) at midspan near the leading edge increased on both the pressure and suction surfaces. Previous midspan measurements as well as computational studies showed that these high frequency oscillations in heat transfer are caused by periodic passing of NGV shock waves across the rotor surface [2,41]. The magnitude of these oscillations increased at lower rotor speeds [1]. The higher temperature at midspan means that locally the rotor is operating at a lower corrected speed (116% versus 120%). Therefore consistency exists between previous and current ac heat flux data.

5.2.3. **Conclusions from radial distortion data**

As stated earlier, influence of temperature distortion on heat transfer can be considered to consist of two mechanisms; the change in driving temperature and change in local heat transfer coefficient. Three methods of predicting gas temperature were evaluated and the resultant Nusselt numbers were compared between distortion and uniform tests. Table 5-1 is a summary of this comparison. The mean effects on the pressure or suction surface for three spanwise locations were obtained by averaging the Nusselt number from leading edge to trailing edge. It should be emphasized that the heat flux data uncertainty is about 5-10%, as was concluded in Chapter 4.
Table 5-1: Nusselt number difference between radial distortion and uniform tests.

<table>
<thead>
<tr>
<th>Method</th>
<th>Midspan</th>
<th>Tip</th>
<th>Hub</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>P.S.</td>
<td>S.S.</td>
<td>P.S.</td>
</tr>
<tr>
<td>Bulk</td>
<td>30%</td>
<td>20%</td>
<td>30%</td>
</tr>
<tr>
<td>2D Streamline</td>
<td>10%</td>
<td>5%</td>
<td>10%</td>
</tr>
<tr>
<td>3-D Euler</td>
<td>10%</td>
<td>5%</td>
<td>10%</td>
</tr>
</tbody>
</table>

The heat flux measurement shows that radial temperature distortion can lead to between 20-30% heat transfer variations, as indicated by the Nusselt number changes based on bulk temperature. At both midspan and the tip, the heat flux increase was inferred to be caused by gas temperature increases. At the tip pressure surface, heat flux changes were not explained by gas temperature estimates from 2D streamline or 3-D Euler method. Instead, midspan gas temperature was found to explain the increased heat flux at the tip pressure surface, implying the influence of the tip leakage flow. Hub suction surface heat flux variations were not explained by gas temperature estimates using either 2D streamline approach or the 3-D Euler method. In particular, 2D streamline approach did a poor job in correlating hub pressure surface heat transfer coefficient between uniform and distortion data.

5.3. Rotor Surface Heat Transfer with Circumferential Distortions

5.3.1. Time-averaged heat flux data

For purely circumferential temperature distortions, i.e., without radial temperature gradient, the hot/cold fluid migration process occurs in the axial-tangential plane of the blade passage. In 2-D, hot gas preferentially collects on the pressure surface, resulting in
higher time-averaged gas temperatures. Similarly, the suction surface gas temperature would be reduced. Combined radial and circumferential distortions (hot spots or hot streaks), would have a combined effect. The hot spot temperature distribution shown in Fig. 4.3 indicates that the variation is mostly along circumferential directions, with a 10% $\theta$TDF factor. The spot center is located roughly at 40% span from the hub, slightly inward of the midspan heat flux gauges (46% span). The circumferentially averaged radial temperature profile has an RTDF factor of 3%.

The heat flux data comparison is made using data from sector A and sector B of a single test, T171, exploiting the higher relative accuracy of the ac heat flux measurements. Because temperature variation is achieved by injecting hot gas into an otherwise uniform flowfield, the averaged inlet enthalpy in the distortion sector is higher than the uniform case. Again the 2D streamline method is opted to normalize heat flux into Nusselt numbers. These results are presented in Fig. 5.27 through Fig. 5.29. The agreement between distorted and uniform flow tests is about 5%. There is a small but consistent difference on the pressure surface at midspan. Because of the gas temperature uncertainties, the Nusselt numbers presented in Fig. 5.27 have better relative accuracy than absolute accuracy. In other words, the chordwise “shape” is more accurate than the absolute level. This implies that the 10% circumferential temperature distortion increased the chordwise heat transfer nonuniformity. The difference between the pressure and suction surface heat flux is larger with distortion than with uniform inlet conditions. The same trend exists on the tip and hub heat transfer data, Fig. 5.28 and Fig. 5.29.

The apparent success of the 2D streamline method in correlating the $\theta$TDF heat transfer data implies, that the hot/cold gas segregation phenomenon with the distortion level tested has a small influence on the heat flux distribution. To further this investigation, an inviscid calculation with the experimental data as input parameters were carried out. Time-averaged surface temperatures around the perimeter of the airfoil where heat flux
gauges were located are shown in Fig. 5.30. For each blade section, the gas temperature is normalized by values at leading edge of same spanwise location. The chordwise temperature variation are also shown in terms of gas-to-metal temperature difference, therefore heat flux influence, on the right hand side vertical axis. Although the general trend of higher pressure surface temperatures than that of the suction surface is visible, the difference, however, only translates into a heat flux variation of 5%, consistent in trend with the experimental data. This effect is small comparing to heat flux data uncertainties of 5-10%. Nusselt numbers calculated using CFD surface temperature are shown in Fig. 5.31 through Fig. 5.33. The agreement is within heat flux data uncertainty levels.

5.3.2. Time-resolved heat transfer

The time-unsteady heat transfer based on spot passing frequency (3 times NGV passing) are presented in Fig. 5.34 and Fig. 5.36 for conditions with and without circumferential distortion, by ensemble-averaging of sector A and sector B data from test T171. There is excellent agreement at the edge of the spot between distortion and uniform tests. Other than the frequency content based on NGV passing, heat transfer responds to the unsteady perturbation caused by temperature distortion. Higher spot temperature leads to higher heat transfer rates. The averaged heat flux increases differ from tip to hub, consistent with the radial temperature variation measured at NGV exit. There is an increase in the ac fluctuations near blade leading edge caused by the hot spot. The time shift in the peak heat flux from leading edge to trailing edge on the pressure surface is evidently associated with the longer convection time (approximately 1/3 of the spot passing period), compared to the shorter convection time on the suction side.

5.3.3. Conclusions from circumferential distortion tests

Although the introduction of hot spot (10% 6TDF with length scale of 1 NGV pitch) caused higher heat transfer rates on the blade surface, the heat flux increase is
attributed to gas temperature increases. *Nusselt numbers based on leading edge gas
temperatures agree with uniform flow data, to within the heat flux data uncertainties of
5-10%.* There is a small but consistent trend that the chordwise heat flux distribution is
more nonuniform with distortion than with uniform inlet conditions. This is consistent in
trend between the experimental data and the 3-D Euler predictions.

For the experimental configurations used herein, the hot gas injector was
operating at the highest temperatures, as imposed by the chemical stability of test gas
component Freon. Time and budget constraints did not permit further experimental work
to increased hot spot (or hot streak) temperatures. Therefore, further analysis of hot
streak influence on turbine rotor heat transfer is continued computationally, using the 3-D
Euler procedure at hand. It is already demonstrated that the introduction of a low level of
temperature distortion (10 % TDF, or hot to cold temperature ratio of 1.1) has a small
effect on heat transfer (less than 5%). How would higher levels of temperature distortion
influence the turbine heat transfer? Through what mechanisms? The next chapter will be
devoted to answer these questions.
Nomenclature

V: absolute velocity

W: relative velocity

U: rotor wheel speed

h&c: hot or cold

Fig. 5.1: Velocity triangles at the NGV rotor interface.

Effects of incidence variation on mean heat transfer rate at the nominal "design" case, $Me = 1.18$, $Re=8.19E5$.

Fig. 5.2: Cascade heat transfer data by Ashworth et al [4].
Fig. 5.3: Midspan heat transfer distribution at design and -10° incidence.

Fig. 5.4: Midspan Nusselt number comparison between 12% RTDF and uniform inflow conditions.
Fig. 5.5: Nusselt number comparison between 12% RTDF and uniform inflow conditions. Tip section (79% span).

Fig. 5.6: Nusselt number comparison between 12% RTDF and uniform inflow conditions at nominal Hub section (16% leading edge span).
Fig. 5.7: Chordwise-integrated pressure surface Nusselt number at three blade sections of hub (16%), midspan (46%) and tip (79%). Reference gas temperature from 2D streamline method.

Fig. 5.8: Chordwise-integrated suction surface Nusselt number at three blade sections of hub (16%), midspan (46%) and tip (79%). Reference temperature from 2D streamline method.
Fig. 5.9: Parabolic NGV inlet temperature profile fit from data.
Fig. 5.10: Snapshot of unsteady static pressure in the blade-to-blade rotor passage at midspan. There is a difference in contour intervals.
Fig. 5.11: CFD Calculated rotor inlet relative flow angle with and without radial distortions.

Fig. 5.12: CFD calculated rotor inlet relative total pressure as function of span.
Fig. 5.13: Rotor relative total temperature contours in the blade-to-blade passage. Hot midspan gas migrates to the pressure surface and spreads out radially toward endwalls. Temperature normalized by NGV inlet average.
Fig. 5.14: Rotor surface streamlines with uniform inflow conditions.

Fig. 5.15: Rotor surface streamlines with 12% RTDF.
Fig. 5.16: Rotor surface total temperature contours from CFD results with uniform inlet temperature. Temperature normalized by NGV inlet average.

Fig. 5.17: Rotor surface relative total temperature contours with 12% RTDF. Temperature normalized by NGV inlet average.
Fig. 5.18: Midspan Nusselt number comparison between 12% RTDF and uniform inflow conditions, using local gas temperature from CFD code.

Fig. 5.19: Nusselt number comparison between 12% RTDF and uniform inflow conditions. Tip section (79% span).
Fig. 5.20: Nusselt number comparison between 12% RTDF and uniform inflow conditions, using CFD calculated surface temperature as reference. Hub section (16% span).

Fig. 5.21: Reproduced from ref [28]. Temperature contours in axial cuts of a rotor passage, showing tip leakage flow draws midspan fluid to tip region.
Fig. 5.22: Tip section pressure surface Nusselt number comparison between 12% RTDF and uniform conditions. Midspan gas temperature used to normalize tip section heat flux.

Fig. 5.23: Surface flow visualization of the same rotor geometry in a transonic cascade wind tunnel [4], Showing endwall secondary flows.
Fig. 5.24: NGV passage ensemble-averaged unsteady heat flux with and without radial temperature distortion. Midspan.

Fig. 5.25: NGV passage ensemble-averaged unsteady heat flux with and without radial temperature distortion. Tip section.
**Fig. 5.26:** NGV passage ensemble-averaged unsteady heat flux with and without radial temperature distortion. Hub section.

**Fig. 5.27:** Midspan rotor Nusselt number comparison between 10% θTDF and uniform case. 2D streamline method.
Fig. 5.28: Tip section rotor Nusselt number comparison between 10% θTDF and uniform case. 2D streamline method.

Fig. 5.29: Hub section rotor Nusselt number comparison between 10% θTDF and uniform case. 2D streamline method.
Fig. 5.30: Heat flux changes due to blade surface temperature variation from leading edge values. CFD calculation with a 10% θTDF.

Fig. 5.31: Midspan Nusselt number comparison between uniform and 10% θTDF. Surface temperature from CFD calculation used as reference temperature.
Fig. 5.32: Tip section Nusselt number comparison between uniform and 10% $\theta$TDF. Surface temperature from CFD calculation used as reference temperature.

Fig. 5.33: Hub section Nusselt number comparison between uniform and 10% $\theta$TDF. Surface temperature from CFD calculation used as reference temperature.
Fig. 5.34: Ensemble-averaged unsteady heat flux data with and without circumferential temperature distortion. Midspan.

Fig. 5.35: Ensemble-averaged unsteady heat flux data with and without circumferential temperature distortion. Tip section.
Fig. 5.36: Ensemble-averaged unsteady heat flux data with and without circumferential temperature distortion. Hub section.
In discussions of the heat transfer data of previous chapter, it was concluded, both experimentally and computationally, that a 10% OTDF temperature distortion had a small influence on rotor heat transfer. The measured effects were quantitatively the same order in magnitude as the heat transfer data uncertainty. The 3D Euler calculation based on the experimental conditions showed that surface temperature changes due to hot gas redistribution in the rotor only represents heat flux variations comparable to data uncertainties, consistent with the experimental results. Here we continue the investigation of temperature distortion on heat transfer, by conducting a numerical experiment using the 3D unsteady Euler solver described in Chapter 3 with higher temperature distortion levels. The focus here is to explore flow phenomena associated with inlet temperature distortion in turbines, with a particular interest in the impact on turbine heat load. It was demonstrated that calculations of the unsteady flow (with relative motion between blade rows modeled) are essential when circumferential temperature distortion is present [55]. This is the approach used here. In this context, an Euler procedure is preferable to a Navier-Stokes code, because of the computer power requirements of the latter. The use of an Euler code means that viscous effects are not modeled. However, as is seen in subsequent sections, important physical insight can still be obtained.
### 6.1. Approach

A brief description of the 3D unsteady Euler code was given in Chapter 3. Details of the numerical procedure can be found in [46]. The fully nonlinear Euler equation set is solved in a domain with 3 stator and 5 rotor passages. This is a close approximation of the 36 by 61 blade count of experimental hardware. The rotor blade is therefore scaled by a factor of 61/60. The computational grid is shown in Figs 3.1 and 3.2. Turbine operating parameters were based on the experimental conditions shown in Table 4-5.

A circular bell-shaped temperature distortion (hot streak) was superposed on top of the otherwise uniform temperature field at the stator inlet, with the total pressure kept uniform. The hot streak is positioned midway between stator leading edges and at 50% span, into one of the three stator passages (see Fig. 6.1). This is equivalent to having 12 hot streaks around the 360° annulus, typical of engine fuel injector placement. A gaussian temperature profile is used with a half height width of 40% of stator pitch. This length scale is comparable to that tested by Butler [5], and allows sufficient grid resolution across the hot streak to avoid numerical dissipation of the temperature gradient. Two levels of temperature distortions were examined, with peak to freestream temperature ratios of 1.4 and 1.8. According to design parameters of the ACE stage, a temperature ratio of 1.4 represents a stoichiometric flame embedded in a uniform flowfield at the averaged turbine inlet temperature. However, turbine inlet temperatures can vary anywhere between stochiometric and compressor discharge levels. Temperature ratios as high as 2 have been measured [5]. For the purpose of exploring flow phenomena, most of the analysis that follows will be based on the calculation with a temperature ratio of 1.8. Comparisons between temperature ratios of 1.4 and 1.8 are used to show scaling.

The unsteady flowfield was obtained through time-marching, and relative motion between the stator and rotor was accounted for using nonreflective boundary conditions at
the stator rotor grid interface. It took about 10 periods (one period being the time for the rotor grid to move across 3 stator passages) to achieve a periodic solution. It should be emphasized that the time-averaged temperature shown here was obtained by direct averaging of the instantaneous temperatures through a complete flow cycle (hot steak period), and not derived from other time-averaged flow variables.

6.2. Description of Flowfield and Hot Streak Migration in the Rotor

One snapshot of the unsteady flowfield is shown in Fig. 6.2. The location of the hot streak (fluid with temperatures above half height), is shown using entropy as marker. One stator airfoil was deleted for a clearer view of the hot streak. Since total pressure is uniform at the stage inlet, the flow pattern in the stator should be independent of temperature distortion, as stipulated by Munk and Prim. The hot streak is periodically chopped by the rotor and is convected downstream within the rotor passage. Because there are five rotor blades for each hot streak, flow conditions in successive rotor passages can also be viewed as events at different times of the flow cycle, Fig. 6.2. The accumulation of high temperature fluid on the pressure surface can be seen by comparing the two rotor passages which have the hot streak within them (see blade #4 and #5 in Fig. 6.2). The hot streak as marked is about half blade height before entering the rotor, whereas on the pressure surface of blade #5 it has spread over the entire span. This preferential migration of hot fluid toward the pressure surface can be explained from simple velocity triangle arguments\(^{1}\), see Fig. 5-1. The hot streak exits the stator at roughly the same Mach number and flow angles, with the high temperature fluid at larger velocity.

\(^{1}\) It is fair to say that this mechanism is well known and have been identified by many authors who visited the subject of inlet temperature distortion in turbines. A detailed explanation was also given by this author in Chapter 5. Therefore it is briefly summarized here.
in the absolute frame. In the rotor relative frame, the hot fluid has a slip velocity toward
the pressure surface. This hot cold gas segregation leads to higher time averaged
temperatures on the pressure surface downstream of the leading edge. The time-averaged
total temperature distribution on the pressure and suction surfaces are given in Fig. 6.3.
Since the differential migration process occurs with both radial and temperature distortion,
the midspan fluid tends to be transported toward the endwalls on the pressure surface and
to be squeezed into the midspan region on the suction surface.

Another observation from Fig. 6.2 and Fig. 6.3 is that the hot streak tends to
move toward the hub, an effect of the buoyancy due to rotation. This radially inward
displacement is distinctive in Fig. 6.4, where the hot streak marked by entropy is seen at
the exit of rotor passage #4. The temperature distribution on the pressure surface also
shows this drift of high temperature fluid to lower radii. A simple analysis using the radial
equilibrium equation will now be used to show the scaling of the buoyancy influence.

6.3. Buoyancy Effects

Radial equilibrium is a balance between the radial pressure gradient and the
centrifugal acceleration of flow swirl. The following expression holds in the stationary
frame of reference

\[
\frac{\partial p}{\partial r} = \rho \frac{V_\theta^2}{r} \frac{1}{R}
\]

where \( p \) and \( \rho \) are pressure and density, \( V_\theta \) is the absolute tangential velocity, \( r \) is radial
axis and \( R \) is the radius at which Eq 6.1 is evaluated. With the introduction of hot streak

\[\text{Eq 6.1}\]

\[\frac{\partial p}{\partial r} = \rho \frac{V_\theta^2}{r} \frac{1}{R}\]

\[\text{Eq 6.1}\]

1 This simple radial equilibrium is valid only when the stream surface is approximately at constant radius.

160
at NGV inlet, the flow pattern inside the stator is unchanged according to Munk and Prim. Although the high temperature fluid has a lower density than the freestream (by a factor of \( T_0/T_{hs} \), where \( T_0 \) and \( T_{hs} \) are the freestream and hot streak temperatures), \( V_0^2 \) of the hot streak fluid is higher by a factor of \( T_{hs}/T_0 \). Therefore, the centrifugal acceleration \( \rho \frac{V_0^2}{R} \) is the same as that without temperature distortion. The hot fluid in the rotor, however, is subjected to the radial pressure field established by the freestream flow, and has a lower centrifugal acceleration because of its lower density and reduced swirl upon impingement with the rotor blade. The imbalance results in a net force that drives the hot streak toward lower radii. This is much easier to understand in the rotor relative frame, where lighter fluid moves in the opposite direction of \( \mathbf{g} \), i.e., the buoyancy influence.

The radial displacement can be estimated using the radial equilibrium equation in rotor relative frame. For simplicity, a force balance on the fluid element from the hot streak core while it glides on the pressure surface will be performed. The pressure gradient is established by the background flow, as denoted by subscript "o"

\[
\frac{\partial p_0}{\partial r} = \rho_0 \Omega^2 R + 2 \rho_0 W_{\theta,0} \Omega + \rho_0 \frac{W_{\theta,0}^2}{R}
\]

Eq 6.2

where \( W_\theta \) is tangential velocity in the rotor relative frame and \( \Omega \) is the rotational speed. The terms on the right hand side of Eq 6.2 represent the centrifugal acceleration due to rotation, the coriolis force, and relative swirl\(^1\). The tangential velocity on the rotor surface, for the ACE rotor, is much smaller than the rotor wheel speed over a large portion of the blade surface, as plotted in Fig. 6.5. Therefore Eq 6.2 can be reduced to the following

\(^1\) All three terms combined will convert to Eq 6.1 if the absolute velocity is used.
\[
\frac{\partial p_0}{\partial r} = \rho_0 \Omega^2 R
\]

Eq 6.3

Denoting radial displacement toward the hub by \( \delta \) and assuming that the static pressure field is approximately that established by the uniform flow, a force balance of the hot streak core fluid yields

\[-\frac{\partial p_0}{\partial r} + \rho_{hs} \Omega^2 R = -\rho_{hs} \frac{D^2 \delta}{Dt^2}\]

Eq 6.4

where \( D^2/Dt^2 \) stands for the Lagrangian derivative. Combining Eq 6.3 and Eq 6.4 and recognizing that \( Dt = dx/W_x \), \( \rho_0/\rho_{hs} \sim T_{hs}/T_0 \), and \( \frac{W_x}{\Omega R} = \phi \), we have

\[
\frac{d^2 \delta}{dx^2} = \frac{1}{R} \left( \frac{T_{hs}}{T_0} - 1 \right) \left( \frac{\Omega R}{W_x(x)} \right) = \frac{1}{R} \left( \frac{T_{hs}}{T_0} - 1 \right) \frac{1}{\phi^2}
\]

Eq 6.5

We can predict the radial displacement of the hot streak core fluid on the pressure surface by integrating Eq 6.5 using the axial velocity distribution plotted in Fig. 6.5. The hot streak core trajectory can also be interrogated from the CFD solution. A comparison between the two is given in Fig. 6.6. The good agreement indicates that the simple analysis outlined above captured the essential physics responsible for the hot streak migration toward the hub.

This simple analysis provides insight about physical scaling of the buoyancy effect. First, the effect scales linearly with hot streak temperature. Second, the dependence on rotational speed (centrifugal force) and residence time (axial velocity) lumps into a familiar nondimensional parameter \( \phi \), the flow coefficient. At low flow.
coefficients (evaluated locally) the effect of buoyancy is more pronounced than at high flow coefficients. For the ACE turbine operating at 120% rotor speed, the averaged flow coefficient at rotor inlet is about 0.5. Because of the high loading of this rotor blade, the local values on the pressure surface are lower than average, see Fig. 6.5.

In addition to the effect on blade surface temperature as shown in Fig. 6.3, buoyancy can affect the gas temperature at the blade root. The time averaged rotor relative total temperature on the rotor platform is shown in Fig. 6.7. The local temperature at the corner between the pressure surface trailing edge and the hub wall is about 5% higher than rotor inlet values for a hot streak temperature ratio of 1.8. For a gas-to-metal temperature ratio of 1.35 (ACE design), this would increase the local heat flux by 20%.

The hot/cold fluid segregation and influence of buoyancy result in temperature nonuniformities on the blade surface and hub wall. How much heat transfer variation does these temperature changes translate into? Here we consider a 2-D blade section across the maximum temperature point shown in Fig. 6.3. The surface temperature around the airfoil perimeter is shown in Fig. 6.8, for two levels of temperature distortion and uniform stator inlet temperatures. For the ACE rotor, the ratio between rotor inlet relative total temperature and the mean metal temperature is about 1.35. The surface temperature variation from leading edge values was converted into heat flux changes by comparing to the mean gas-to-metal temperature difference. For a hot steak temperature ratio of 1.4, the peak local heat flux change on the pressure surface is about 10%. However, as the distortion level is doubled, the change in heat flux is about 40%. This nonlinear influence is qualitatively explainable using the velocity triangle arguments. The increase in averaged surface temperature is proportional to the temperature of the accumulated fluid, and the accumulation rate, as indicated by the slip velocity in Fig. 5.1, scales with hot streak temperature as well.
6.4. Hot Streak Redistribution In the Stator-Rotor Gap

The flow field in Fig. 6.2 shows the unsteady time and length scale associated with hot streak/rotor interaction. Unsteady simulations retaining the relative motion between hot streak and rotor are necessary to capture the physics of flow segregation in the rotor passage. At the leading edge, the rotor sees the hot streak peak temperature periodically. The time-averaged temperature has the most influence on turbine durability. It is commonly believed that the relative motion between hot streak and rotor smears out the pitchwise temperature gradient; that the time-averaged total temperature relative to the rotor is circumferentially uniform at the rotor inlet; that only the radial temperature gradient is retained. We will show here that circumferential variations do occur in the time-averaged relative total temperature at rotor inlet. The time-averaged flowfield at two axial planes upstream of the rotor (leading edge and 20% chord upstream) are shown in Fig. 6.9 and Fig. 6.10a, for a hot streak temperature ratio of 1.8. The rotor not only sees the periodic passing of the hot streak, it is also subjected to an azimuthal as well as radial variation in the time-averaged temperature. The radial variation is understandable because of the radial temperature gradient of the hot streak. The pitchwise variation, present even 20% chord upstream of rotor leading edge, is less intuitive. It was the observation of this variation with a length scale of the rotor pitch, that prompted the following investigation.

The implications of this pitchwise variation are significant. Not only does it mean that the blade leading edge sees a time-averaged temperature different from the azimuthal mean, but also it implies a new flow phenomenon that is not appreciated at this point. For a hot steak temperature 1.8 times that of the freestream, the circumferential time-averaged temperature variation at midspan as shown in Fig. 6.10b is about 5% of the averaged rotor inlet stagnation temperature, or 200°F for a modern turbine with inlet gas temperature of 3000°F. This temperature variation is significant in magnitude in terms of rotor life. What
causes this temperature variation? Is it a coincidence of this particular turbine design that the blade leading edge temperature is lower than the pitchwise average? How is it dictated by the flow physics? We seek to answer these questions here.

A separate CFD procedure was used to further identify the cause for such circumferential variation. UNSFLO [20], a 2-D unsteady Euler solver was adapted to simulate the stator-rotor flowfield with a hot streak, Fig. 6.11. The blade geometry and boundary conditions were all based on midspan results of the 3-D calculation. Our objective is to see whether a 2-D approach would produce circumferential variations similar to the 3-D results. Such a comparison is given in Fig. 6.11 b), where time-averaged rotor relative total temperatures are plotted against pitchwise positions at a location 20% chord upstream of rotor leading edge. The agreement between 3-D and 2-D results is excellent, both in phase and magnitude. This implies that the flow phenomenon is basically two dimensional.

As the hot streak approaches the rotor blade, (Fig. 6.2 and Fig. 6.11a), it responds to the potential field of the rotor, twisting and deforming as it impinges with blade leading edge. If the potential interaction between the hot streak and rotor contributes to the azimuthal temperature variation, the interaction would attenuate as the upstream distance to the rotor was increased, since the potential influence of the rotor decays exponentially with axial distance. One way of investigating this is to perform a rotor-only calculation, where an unsteady hot streak is specified at the rotor inlet flow boundary. Again UNSFLO was used for this calculation, as illustrated in Fig. 6.12. Inflow conditions were specified according to stator exit flows from the 2-D calculation. The hot streak temperature profile is the same as that used in the 2-D stator-rotor calculation. Time averaged rotor relative total temperature as a function of pitchwise position is shown in Fig. 6.12 b), together with 2-D and 3-D stator-rotor results. Although a small pitchwise variation does exist, it is less than 10% in magnitude than the stator-rotor calculations.
This implies that the potential hot streak/rotor interaction is not the dominant driving mechanism.

A comparison between Fig. 6.11 and Fig. 6.12 implies that the unsteady interaction between the stator and rotor may be the driver for causing the pitchwise variation of time-averaged rotor relative total temperature at the rotor inlet. As a result of this interaction, both the stator and rotor experience the unsteadiness of the other row at frequencies of blade passing. After thorough examination of the flow field, including flow animation using flow visualization tools [26], the unsteady fluctuation of the stator exit flow angle as the rotor passes by was identified as the driving mechanism.

For a hot streak temperate ratio of 1.8, the time history of stator exit tangential flow angle at a point midway between the stator trailing edges is plotted in Fig. 6.13. The same flow angle with uniform inlet conditions is also shown for comparison. First, the flow angle changes at the rotor passing frequency. The level of variation is about 4 degrees. Second, the hot steak does not significantly alter this flow angle variation, suggesting that the stator loading change due to blade row interaction is not strongly affected by the temperature distortion. The change in flow angle causes the hot streak to wobble in the tangential direction, at the rotor blade passing frequency. Two extreme positions of the hot streak are shown in Fig. 6.14 at midspan (again the hot steak was marked with entropy at about half height width). The wobbling of the hot streak means that the passing speed between the hot streak and rotor varies. When the hot streak is moving at a lower speed at certain position, the time-averaged temperature at that position would be higher because of the longer hot streak residence time. On the other hand, if the hot streak is moving faster across another point, that location would sample less hot fluid and resulting in lower time-averaged gas temperatures. The time history of the instantaneous relative total temperatures at two points, corresponding to maximum and minimum averaged temperature locations shown in Fig. 6.10, are shown in Fig. 6.15. One
trace is shifted in time to allow width comparison. The change in hot streak width is a clear manifestation of the mechanisms just described.

Since the flow angle change is caused by the interaction of stator and rotor and is independent of temperature distortion, the hot streak wobble should be independent of the hot streak temperature. The different hot streak "width" as seen at different locations should be independent of hot streak temperature as well. Indeed, this is confirmed in Fig. 6.16 where the temperature history at the same positions as in Fig. 6.15, is given for a hot streak temperature ratio of 1.4. This suggests that the pitchwise variation in relative gas temperature upstream of the rotor leading edge should be linearly proportional to hot streak strength. The magnitude of the variation is a fixed proportion of the extra thermal energy contained in the hot streak. The linear proportionality is shown in Fig. 6.17, where the maximum-to-minimum difference in averaged temperature at midspan is plotted against hot streak temperature. The calculation with uniform inlet conditions is also shown.

For the calculation with uniform inlet conditions, the relative temperature field upstream of the rotor is nonuniform, although the magnitude of variation is negligible comparing to the variations when temperature distortion is present. The variation with uniform inlet flows can be explained by the unsteady loading of the stator, and previously revealed in the analysis of axial compressor flows [47]. Viewed from the rotor frame of reference, the stator can be perceived as a turbine extracting energy from the working fluid. Caused by unsteady interaction between the stator and rotor, the loading of the stator fluctuates, as it passes each of the rotor blades. The work extracted by the stator thus varies, depending on its relative position with the rotor blade. When the stator load is less than average (extracting less work), the thermal energy behind it is higher. On the other hand, when the stator load is more (extracting more work), the thermal energy behind it is lower. On the average, this leads to a pitchwise variation in the relative total
temperature as well as relative total pressure upstream of the rotor.

The nonuniformity in time averaged total temperature upstream of the rotor implies that the rotor leading edge can operate at temperatures different from the circumferential mean. For a hot streak temperature ratio of 1.8, the temperature variation levels indicated in Fig. 6.10 translates into a gas temperature variation of 200°F for a modern turbine with turbine inlet temperature of 3000°F. For the flow conditions examined here, the blade leading edge temperature is lower than the circumferential average (Fig. 6.9), an advantage for lower turbine heat load.

Many design parameters can influence the unsteady blade row interaction process, and thus modifying the time-averaged rotor inflow temperatures. Here we explore one design parameter that can be varied in turbine design, the stator rotor pitch ratio (or relative blade count ratio). The designed blade count is 36 vanes and 61 rotor blades. Calculations discussed above used a blade count of 3 vanes and 5 rotor blades, therefore closely simulates the pitch ratio of the turbine tested. Calculation with a lower blade count ratio of 3 vanes and 4 rotor blades was carried out using the 2-D CFD code (UNSFLO). The rotor blade was scaled by a factor of $\frac{5}{4}$ to maintain the solidity as originally designed. The NGV rotor gap size was kept the same. The stator geometry and inlet hot streak pattern were the same as in the 3:5 blade count. Hot streak to mean flow temperature ratio of 1.8 was used. The time-averaged rotor relative total temperature distribution at the rotor leading edge plane for two different pitch ratios are shown in Fig. 6.18. The magnitude of the temperature variation is a function of relative stator-rotor blade count. For a stator-rotor blade count ratio of 3:4, the magnitude of pitchwise variation is larger than for 3:5. For higher pitch ratios (more rotor blades), the influence of blade row interaction is expected to be less. As the pitch ratio approaches infinity, the influence of rotor unsteadiness vanishes. The rotor blade row becomes an actuator disk. The phase angle (peak temperature location relative to blade leading edge) changes with
blade count as well. For the 3:4 case, the rotor is located at the highest temperature point (to be avoided). A quantitative explanation of this phase change with pitch has not yet been formulated. One speculation could be made through the unsteady shock structure, see Fig. 5.10. As the blade size and pitch change, the angle and position of the reflected shock waves vary, thus changing the phase of NGV load fluctuation. The incentives for further research in this area is obvious. It is perceivable that design parameters can be optimized in such a way that the blade be positioned at the low temperature region.

6.5. Combined Effects

The time averaged blade to blade relative total temperature in 5 rotor passages is shown in Fig. 6.19 at different axial stations from leading edge to trailing edge. The solution is obtained from 3-D calculations with a hot streak to mean flow temperature ratio of 1.8, and blade count ratio of 3:5. All three effects of temperature distortion are apparent. The time-averaged rotor relative total temperature varies circumferentially upstream of rotor leading edge, caused by hot streak wobble as a result of blade row interaction and stator exit flow angle fluctuations. At leading edge plane, time averaged relative total temperature is higher in the passage than at the blade leading edge (also see Fig. 6.9). Increase in pressure surface temperature is caused by hot fluid accumulation onto the pressure surface, as seen from 20% chord to rotor trailing edge. Buoyancy effects drive high temperature fluid on the pressure surface toward the hub region and eventually on to the hub wall, Fig. 6.19 (d) and (e).

It is interesting to noted that, on a “time-averaged” basis, the high temperature fluid formed at the rotor leading edge as a result of NGV-rotor interaction and hot streak wobble, stays detached from the blade surface and is “convected” through the rotor passage.
6.6. Summary

Three separate mechanisms have been identified which influence blade heat transfer when turbine is subjected to hot streak type of distortions. The first two were previously reported in the literature while the third constitutes an increase in our understanding of turbine flow physics. Parametric scaling of these effects with turbine design variables were examined. These phenomena are as follows.

1. Accumulation of hot fluid on the pressure surface increases the time averaged local surface temperatures. The increase in surface temperature depends nonlinearly on the level of temperature distortion. For a hot-to-cold temperature ratio of 1.4, this temperature increase corresponds to a local heat flux increase of 10%. While with a hot-to-cold temperature ratio of 1.8, the change in heat flux is about 40%.

2. The influence of buoyancy drives high temperature fluid toward the hub and increases local platform surface temperature. The radial displacement is proportional to hot streak temperature and inversely proportional to flow coefficient. For a hot streak temperature ratio of 1.8, the radial displacement of the hot streak core is about 50% blade span for the geometry studied here.

3. Unsteady blade row interaction and the resultant NGV exit flow angle fluctuation causes the hot streak to wobble, at the rotor blade passing frequency. This mechanism produces nonuniformity in the time-averaged rotor relative total temperature at rotor inlet. As a result, rotor leading edge temperature is different from the local circumferential mean. For a hot streak temperature ratio of 1.8, this variation at midspan is about 5% of the averaged rotor inlet relative temperature, or 200°F for a modern turbine with inlet gas temperature of 3000°F. The degree of pitchwise variation is linearly proportional to the magnitude of temperature distortion, and is dependent on the relative
stator rotor blade count. The hot streak wobble is a result of the blade row interaction which is not influenced by temperature distortion. This unsteady flow feature is expected to exist in all situations in which strong blade row interaction occurs. NGV coolant wakes, endwall vortices and tip leakage flows, for example, could similarly behave.
Fig. 6.1: NGV inlet temperature distortion in the form of a circular hot streak.
Fig. 6.2: Snapshot of flowfield showing convection of hot streak through turbine stage.
Fig. 6.3: Time averaged total temperature distribution on rotor's pressure and suction surfaces. Hot streak temperature ratio of 1.8. Temperature normalized by midspan leading edge value with contour intervals of 0.02.
Fig. 6.4: Snapshot of rotor exit flow field (looking upstream) showing position of hot streak.
Fig. 6.5: Time-averaged chordwise velocity distribution on rotor pressure surface. Extracted from midspan under uniform NGV inlet conditions.

Fig. 6.6: Simple model predicted and CFD calculated hot streak core trajectory on rotor pressure surface. Mean surface streamline from uniform calculation.
Fig. 6.7: Time-averaged rotor relative total temperature distribution on rotor platform. Temperature normalized by averaged value at hub inlet.
Fig. 6.8: Surface gas temperature distribution and resulted heat flux change, with hot streak to mean flow temperature ratio of 1.4 and 1.8.
Fig. 6.9: Time-averaged rotor relative total temperature at rotor's leading edge plane. Contour intervals are about 2% of averaged rotor inlet relative temperature.
Fig. 6.10: a) Time-averaged rotor relative total temperature at axial station 20% rotor chord upstream of leading edge. b) Midspan pitchwise variation from a).
Fig. 6.11: a) Hot streak calculation using 2-D Euler solver based on midspan flow conditions of 3-D calculation. b) Pitchwise variation of time-averaged rotor relative total temperature 20% rotor chord upstream of leading edge.
Fig. 6.12: a) Rotor-only calculation simulating unsteady hot streak. Inlet conditions based on 2-D stator-rotor calculation. b) Pitchwise variation of time-averaged rotor relative total temperature 20% rotor chord upstream of leading edge.
Fig. 6.13: NGV exit flow angle history.
Fig. 6.14: Two extreme positions of the hot streak showing hot streak wobble. Entropy used as marker.
Fig. 6.15: Time history of rotor relative total temperature at two pitchwise positions with maximum and minimum time-averaged values. Temperature normalized by freestream value. Traces are time shifted for comparison.

Fig. 6.16: Time history of rotor relative total temperature at two pitchwise positions. Hot streak to mean flow temperature ratio of 1.4.
Fig. 6.17: Peak-to-peak magnitude of pitchwise variation of time-averaged rotor relative total temperature at midspan with hot streak temperatures of 1.4 and 1.8. Temperature normalized by circumferential average.

Fig. 6.18: Time-averaged rotor relative total temperature at leading edge plane. Calculation with stator rotor blade count of 3:4 and 3:5. Hot streak temperature ratio of 1.8.
Fig. 6.19: Planar cuts through rotor flow field showing time-averaged relative total temperature. Contour intervals of 2% rotor inlet averaged temperature.
Chapter 7.
Concluding Remarks

The influence of inlet temperature distortions typical of combustor exit flows on turbine heat transfer was investigated both experimentally and computationally, using a highly loaded transonic turbine as the test article. This investigation is summarized here. Major conclusions and lessons learned are outlined. Areas worthy of further pursuit are noted.

7.1. Thesis Summary

The experimental work was carried out at the MIT blowdown turbine facility. The facility was extensively modified to allow detailed aerodynamic and heat transfer measurements with inlet temperature nonuniformity. In particular, aerodynamic survey capabilities using circumferential rake translators were added both upstream and downstream of the turbine stage. This aero survey capability was essential in defining tunnel conditions under which the heat flux data was taken, such as NGV exit temperature distribution.

Aerodynamic data was taken in conjuncture with heat flux measurements. Detailed mapping of the temperature field in the test section, in particular the NGV exit temperature survey for both radial and circumferential temperature distortions, was an essential part of this investigation. This allowed accurate interpretation of heat flux data and determination of the temperature distortion influence.
The acquisition and reduction of rotor heat transfer data were a crucial step in this investigation. Heat flux was measured on the blade surface at three sections representing nominally tip, midspan and hub radii. The data reduction, consistency check, and error analysis of heat flux data using various methods determined and validated data uncertainty as 5-10% in terms of Nusselt numbers. This heat transfer data, together with the temperature measurements, forms a well-defined turbine heat transfer data set detailing the influence of temperature distortion.

Three types of inlet distortions were introduced and their impact on turbine heat transfer measured. These are:

- Momentum turbulence generated from a perforated grid. The turbulence spectrum at the NGV inlet was determined to be comparable to common combustor exit flows (Tu=8%).

- Circumferentially uniform, radially varying temperature profile skewed toward the hub. The NGV exit temperature distribution (at the rotor leading edge location) was determined using the downstream translator rake. The temperature profile had a radial temperature distortion factor of 12% (see definition of RTDF in chapter 1).

- Circumferential temperature distortion through hot gas injection upstream of the stage. The temperature pattern at the NGV exit was measured as 10% ΘTDF with a half height length scale of approximately one stator pitch.

The distortion data were analyzed and interpreted using analytical and numerical methods, including an unsteady 3-D Euler solver. The role of fluid redistribution in the rotor was examined. The conclusions of this analysis and a computational study with higher circumferential distortion levels are given below.
7.2. Conclusions

The distortions tested were found to have the following influence on rotor heat transfer:

- The introduction of momentum turbulence (Tu=8%) at stage inlet was found to have no influence on rotor heat transfer in this fully scaled rotating stage environment.

- A 12% radial temperature distortion (RTDF) increased rotor blade heat transfer by 20-30%, except on the hub suction surface where a reduction of 20% in heat flux was observed. At midspan, the heat flux increase was inferred to be caused by the local inlet gas temperature. Rise in suction surface heat flux at the tip was caused by local inlet gas temperature increase as well. The heat transfer variations at the tip pressure surface and hub section were not explained by the inlet gas temperature changes. Examination of heat flux distribution implies influence of secondary flows. 3-D Euler calculations based on experimental data showed secondary flow influence on the hub pressure surface, but did not explain the measured influence at the tip or hub suction surface. It was found, however, that the tip pressure surface data correlate well with midspan gas temperature, suggesting migration of midspan fluid to the tip region.

- A 10% circumferential temperature distortion (θTDF) was found to have a small influence on blade heat transfer. Chordwise heat flux distribution was less uniform with circumferential temperature distortion than with uniform inlet conditions, with higher heat transfer levels on the pressure surface than the suction surface. 3-D Euler calculations predicted higher pressure surface
temperatures than the suction surface, therefore consistent in trend with the experimental data.

Further computational investigations using higher levels of temperature distortion in the form of hot streaks revealed novel flow phenomena that were not previously identified. There are three mechanisms in which an inlet temperature distortion can influence the rotor blade surface temperature (heat flux).

1. Preferential migration of hot/cold fluid leads to higher time-averaged local surface temperature nonuniformity between the pressure and suction surfaces. This effect scales nonlinearly with temperature distortion amplitude. This nonlinear dependence is qualitatively explainable through kinematic (velocity triangle) arguments.

2. The influence of buoyancy drives high temperature fluid (with lower density) toward the hub. The radial displacement is proportional to hot streak temperature and inversely proportional to local flow coefficient.

3. Unsteady blade row interaction causes the NGV exit flow angle to fluctuate and the hot streak to wobble, at the rotor blade passing frequency. Viewed in the rotor relative frame, the hot streak passes the rotor with different speeds at different positions, thus varying the hot streak residence time. This produces nonuniformity in the time-averaged rotor relative total temperature at rotor inlet. As a result, rotor leading edge temperature is different from the circumferential mean. This influence is linearly proportional to hot streak amplitude, and is a function of at least one design variable, stator-rotor pitch ratio.

The contribution of this thesis contains: 1) the first well-defined turbine rotor heat transfer data with inlet temperature distortions; 2) the determination of the influence of inlet turbulence on rotor heat transfer; 3) detailed analysis of the heat transfer data and discussion of flow mechanisms contributing to the observed distortion influence on heat
transfer; 4) the elucidation of flow phenomena and parametric scaling relating to the influence of inlet temperature distortion on rotor heat transfer.

7.3. **Recommendations for Future Work**

The following items would bear fruit in the near future.

As stated in Chapter 5, viscous flow simulation based on the RTDF data would shed light on the influence of tip leakage flow and endwall secondary flows. The capability of the CFD code on endwall flow modeling should be examined and verified. Relative importance of flow mechanisms can be evaluated to help identify areas where design can be guided to offset or insensitize the influence of temperature distortion.

A series of numerical experiments can be performed on the effects of hot streaks. The phase relationship discussed in chapter 6 should be examined. Can a turbine be designed such that the blade is positioned in a low temperature region? Viscous calculations can be performed to examine relative importance of inviscid and viscous effects.

More heat transfer data from the blowdown turbine facility with higher hot streak temperatures to examine scaling as suggested in Chapter 6. Gas temperature measurements should be taken to allow detailed heat flux data analysis. Measurement of flow variables such as flow angle, total pressure can elucidate the secondary flow structure in the turbine. This would also provides data for code validation and calibration.
References


8 D'Hoop, Emmanuel, Private communication.


19 Epstein, A.H., Private communication.


21 Gleixner, A., Private Communication.


25 Guenette, G.R., Private communication.


43 Roback, R.J. and Dring, R.P.,
A.1. Introduction

Turbine inlet total temperature is a critical quantity in rotor heat transfer data reduction and analysis. With hot gas injection, temperature and mass flow of both the mainstream and injected flow are required to determine the mass averaged temperature into the turbine stage. Main flow temperature is measured by the total temperature probe on the upstream translator, while the injected flow temperatures are measured by four stationary thermocouple probes positioned at the center of the injector exit. Injector mass flow measurement is done using calibrated orifice plates, installed inside each injector body as shown in Fig. 2.6. It is desirable to have similar flow properties among the injectors installed in the 120° annular sector (sector A). This mass flow calibration process also checks and verifies the degree of similarity among the four injectors.

The accuracy requirement of this mass flow measurement can be demonstrated through the following uncertainty analysis. The mass averaged inlet enthalpy with hot gas injection can be calculated using the following equation,

\[ T_{T,\text{avg}} = \frac{\left( m_0 - \dot{m}_{\text{inj}} \right) C_{p,0} T_{T,0} + \dot{m}_{\text{inj}} C_{p,\text{inj}} T_{T,\text{inj}}}{\dot{m}_0 C_{p,\text{avg}}} \]

Eq A.1

where \( T_{T,\text{avg}} \) is the mass averaged inlet total temperature, \( m_0 \) and \( \dot{m}_{\text{inj}} \) are the
mainstream and injected mass flow, $T_{T,0}$ and $T_{T,\text{inj}}$ are temperatures, $C_{p,0}$, $C_{p,\text{inj}}$ and $C_{p,\text{inavg}}$ are the constant pressure specific heat at different temperature levels. Upon differentiation and neglecting the variation of $C_p$ against temperature, Eq A.1 becomes

$$\frac{\delta T_{T,\text{inavg}}}{T_{T,\text{inavg}}} = C_1 \left( \frac{\delta \dot{m}_0}{\dot{m}_0} - \frac{\delta \dot{m}_{\text{inj}}}{\dot{m}_{\text{inj}}} \right) + C_2 \frac{\delta T_{T,0}}{T_{T,0}} + C_3 \frac{\delta T_{T,\text{inj}}}{T_{T,\text{inj}}}$$

$$C_1 = \frac{\dot{m}_{\text{inj}}}{\dot{m}_0} \left( 1 - \frac{T_{T,\text{inj}}}{T_{T,0}} \right) \left( 1 - \frac{\dot{m}_{\text{inj}}}{\dot{m}_0} \right) + \frac{\dot{m}_{\text{inj}} T_{T,\text{inj}}}{\dot{m}_0 T_{T,0}}$$

$$C_2 = \left( 1 - \frac{\dot{m}_{\text{inj}}}{\dot{m}_0} \right) + \frac{\dot{m}_{\text{inj}} T_{T,\text{inj}}}{\dot{m}_0 T_{T,0}}$$

$$C_3 = \frac{\dot{m}_{\text{inj}}}{\dot{m}_0} \frac{T_{T,\text{inj}}}{T_{T,0}} \left( 1 - \frac{\dot{m}_{\text{inj}}}{\dot{m}_0} \right) + \frac{\dot{m}_{\text{inj}} T_{T,\text{inj}}}{\dot{m}_0 T_{T,0}}$$

Eq A.2

For the highest injection temperature of 670°K (limited by Freon-12, to about 700°K) and average freestream temperature of 480°K, The temperature ratio of $T_{T,\text{inj}}$ and $T_{T,0}$ is ~ 1.4. With four injectors installed in a 120° sector, each injector covers 1/12 of the annular flow area, approximately 15.5 in$^2$ with outer and inner wall diameters of 21.98" and 15.68". The injector exit flow area is 1.021 in$^2$ (diameter of 1.14"). For parallel flows with the same total pressure (thus same Mach number), the mass flow is proportional to area and inversely proportional to the square root of the flow total temperature. Therefore, the mass flow ratio is approximately 0.06. Calculating the influence coefficients in Eq A.2 and assume that the uncertainty sources are not correlated, then the net uncertainty of the
The averaged total temperature is

\[
\frac{\delta T_{T,\text{avg}}}{T_{T,\text{avg}}} = \sqrt{0.023^2 \left( \frac{\delta \dot{m}}{\dot{m}_0} \right)^2 + \left( \frac{\delta \dot{m}_{\text{inj}}}{\dot{m}_{\text{inj}}} \right)^2 + 0.92^2 \left( \frac{\delta T_{T,0}}{T_{T,0}} \right)^2 + 0.082^2 \left( \frac{\delta T_{T,\text{inj}}}{T_{T,\text{inj}}} \right)^2}
\]

Eq A.3

Uncertainty of mass flow measurements of the order of 10% contribute to the averaged total temperature error by 0.3%, or about 2°C, which is acceptable for heat transfer data reduction. Because of the fraction of mass flow injected is very small, the injected gas temperature uncertainty contributes little to the mass-averaged gas temperature uncertainty.

### A.2. Calibration Method

Most mass flow measurements are made through the use of a flow resistance device, such as an orifice plate. The pressure loss through such a device is proportional to the dynamic head of the flow. Mass flow is related to the pressure drop, fluid density and the effective flow area of the orifice. If the flow Mach number at the orifice is high (greater than 0.3), compressibility effect becomes important. In such a case, the mass flow is also dependent on the ratio of specific heats. The pressure loss mechanisms through such a device is solely inertial, therefore knowledge of the fluid viscosity is not critical.

Fig. A.1 illustrates the injector mass flow calibration setup. Supply gas enters a rotameter, throttle valve and the entire hot gas injection system. An orifice plate is located inside the injector body, between the right angle flow entrance and honeycomb. Unlike standard orifices, a perforated plate is used to ensure uniform flow at the injector exit. The perforated plate consists of 1/16" diameter holes on a rectangular pattern, with center-to-
center spacing of 0.110". The hole diameter is chosen to be large enough to avoid flow Reynolds number dependence.

The pressure drop across the orifice plate is determined through pressure measurements at two easily accessible locations, Fig. A.1. The upstream pressure is measured downstream of the tube bundle heat exchanger. The orifice exit pressure is taken as the injector exit pressure. For calibration, the injector exit static pressure is the room atmospheric pressure. During a blowdown test, this pressure is taken as that measured by the stationary total pressure probe located in sector A, see the instrumentation section of Chapter 2. For very low Mach numbers (~0.07), the total to static pressure difference is only 0.3% of the absolute level, less than the accuracy of the pressure transducers. Therefore using total rather than static pressure is acceptable for injector mass flow calculation. The injector exit temperature is measured by a stationary type K thermocouple probe. This temperature is assumed to be the flow temperature through the orifice plate.

Mass flow depends on the effective flow area of the orifice plate, which can be determined if another independent mass flow measurement is given. A commercially available Fischer-Porter rotameter (Model: 10A3555A) is chosen, Fig. A.1. For a glass tube number of FP-1-27-G-10 and float number of 1-GSVGT-64-T6, the full scale capacity of the flow meter is 17.5 SCFM air under standard atmospheric conditions (14.7 PSIA and 70°F by Fischer Porter Definition). This rotameter is an uncalibrated type, meaning that the accuracy is 2% of the full scale value. The accuracy of this rotameter was compared with the flow meter used for filling argon gas into the supply tank, and the agreement between the two was about 2%.

The mass flow through a rotameter depends on the pressure, temperature and the molecular weight of the test gas. The rotameter inlet temperature is measured by a type K thermocouple while the pressure is measured by the MKS baratron located on the BDT’s main control panel. The mass flow can be calculated using the following equation
\[ \dot{m} = \phi \dot{m}_{\text{fullscale}} \sqrt{\frac{M_{\text{gas}}}{M_{\text{air}}} \frac{P_{\text{Rotameter}}}{P_{\text{STD}}} \frac{T_{\text{STD}}}{T_{\text{Rotameter}}}} \]

Eq A.4

where \( \phi \) is the percentage reading from the rotameter, \( \dot{m}_{\text{fullscale}} \) is the full scale capacity (0.00992 Kg/s) under standard conditions (\( P_{\text{STD}} = 14.7 \text{PSIA} \) and \( T_{\text{STD}} = 294.3^\circ \text{K} \)).

\( P_{\text{Rotameter}} \) and \( T_{\text{Rotameter}} \) are the pressure and temperature measurements from the rotameter inlet. If the test gas is not air, a correction is made using the test gas molecular weight \( M_{\text{gas}} \).

A.3. Calibration Results

Baseline calibration was carried out by blowing air in steady state. Pressures upstream and downstream of the orifice plate are measured by the BDT baratron. With the mass flow measurement from the rotameter, data can be fit into the standard compressible orifice equation to determine the effective flow area of the orifice plate. If \( \dot{m}_{\text{inj}} \) is mass flow, \( T_T \) is total temperature at the injector exit, \( P_1 \) and \( P_2 \) are upstream and downstream pressures across the orifice, \( R \) and \( \gamma \) are the gas constant and ratio of specific heats respectively, then the orifice equation can be written as

\[ \frac{\dot{m}_{\text{inj}} \sqrt{R_T}}{P_1} \sqrt{\frac{R}{\gamma}} = A \left[ \frac{2}{\gamma - 1} \left( \frac{P_1}{P_2} \right)^{\frac{\gamma - 1}{\gamma}} - 1 \right] \left( \frac{P_1}{P_2} \right)^{-\frac{\gamma + 1}{2\gamma}} \]

Eq A.5

where \( A \) is the effective flow area of the orifice plate. However, a least squares fit between data and Eq A.5 revealed a systematic error which is proportional to the pressure.
\[
\Delta P = \frac{P_1 - P_2}{P_1}. \text{ Thus a modified orifice equation is used as the following,}
\]

\[
\frac{\dot{m}_{\text{inj}} \sqrt{TR}}{P_1} \sqrt{\frac{R}{\gamma}} = A \sqrt{2 \left( \frac{P_1}{P_2} \right)^{\gamma-1} \left( \gamma - 1 \right)} \left( \frac{P_1}{P_2} \right)^{-\frac{\gamma+1}{2\gamma}} \left( 1 + \alpha \frac{\Delta P}{P_1} \right)
\]

\text{Eq. A.6}

The above equation with two values of \(\gamma\) is plotted as solid lines in Fig. A.2. Left hand side of Eq. A.6 is plotted as a function of the pressure ratio \(P_1/P_2\). The open circles are the baseline steady state calibration data, through which the effective flow area \(A\) was determined by a least squares fit. The value of \(\alpha\) is also determined. For a typical blowdown turbine test, the pressure ratio across the orifice plate is about 1.3, well within the range the calibration was carried out.

Given the complexity of the flow and the semi-empirical term in the orifice equation, the validity of Eq. A.6 was extensively tested with all possible flow conditions encountered during a typical blowdown turbine test, such as different gas temperatures and flow rates. Tests at high temperatures were carried out by heating the tube bundle heat exchanger and the flowpath of the injector. For gas temperatures higher than room temperature, data were taken during the first second of a blowdown, after the initial transient and before the gas temperature starts to drop. Because it is difficult to use the turbine test gas mixture of Argon and Freon (\(\gamma = 1.28\)) for calibration, carbon dioxide (\(\text{CO}_2, \gamma = 1.26\)) was used as a substitute to verify the dependence on \(\gamma\).

As can be seen from Fig. A.2, the measured mass flow follows that predicted from Eq. A.6 very well. The percentage variation between the measured and predicted is plotted in Fig. A.3. It can be concluded that the mass flow measurement technique is repeatable to within ±2%. Taking into account of the systematic error of the uncalibrated rotameter (2%), the mass flow measurement uncertainty should be within 3-4%, accurate.
enough for turbine inlet total temperature measurement calculations.

The careful calibration and verification described above was done on one of the four injectors only. With the technique verified, the other three injectors were only calibrated in steady state flow conditions using air. Table A-1 lists the final results of these calibrations. The injectors are ordered such that injector one is located at -45° position, see Fig. 2.7.

Table A-1: Calibrated effective flow area of orifice plates used in Eq A.6. \(\alpha = 0.346\)

<table>
<thead>
<tr>
<th>Injector Number</th>
<th>Orifice Area, (10^{-5})M²</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>8.74</td>
</tr>
<tr>
<td>2</td>
<td>8.34</td>
</tr>
<tr>
<td>3</td>
<td>8.48</td>
</tr>
<tr>
<td>4</td>
<td>8.45</td>
</tr>
</tbody>
</table>

A note about the orifice inlet and exit pressure measurements used in deducing injector mass flow during a turbine blowdown test. These pressure measurement transducers (Kulites) are calibrated using a two point method, with the back reference pressure varied from vacuum to atmosphere. The underlying assumption of this method is that the transducers behave linearly with pressure variations. To quantify the error associated with this assumption, the test section was filled with air at various pressure levels and the Kulite transducers were calibrated against the MKS baratron at several points. This calibration indicated that the nonlinearity errors are significant for those transducers used in the orifice inlet pressure measurement, but negligible for the pressure measurement used as the injector exit pressure (< 0.5%). Based on this multiple point calibration, a second order correction can be made to account for transducer nonlinearity. The orifice inlet pressure data were reduced with this second order correction. The two pressure measurements for injectors #2 and #3 agreed to within 0.5%, a good indication of repeatability among the injectors. Mass flow calculation of injectors #1 and #4 used the averaged orifice inlet pressure from injectors #2 and #3.
Fig. A.1: Schematic of the injector mass flow calibration arrangement.
Fig. A.2: Orifice plate mass flow characteristics. Corrected mass flow vs. pressure ratio.

Fig. A.3: Data scatter of injector mass flow using calibrated orifice plate as flowmeter.
Appendix B.
Error Analysis of Heat Transfer Data

In this appendix, the uncertainty levels of time-averaged heat flux data are discussed. The technique is similar to that used by Abhari [1], although details differ somewhat.

As stated in Chapter 4, most of the heat transfer data uncertainty was attributed to the abrupt resistance changes (or jumps) of the heat flux gauge thermometers. After the post-run gauge temperature calibration, the abnormal resistance changes (if any) resulted in differences between the "indicated" initial top and bottom sensor temperatures. This initial temperature difference between the top and bottom sensors becomes an indicator of the accuracy of the gauge temperature calibration, thus an indicator of the dc heat flux uncertainty.

The dc heat flux is related to the gauge sensor temperature by the following expression

\[ \dot{Q} = \frac{k}{d} (T_t - T_b) \]

Eq B.1

where \( \dot{Q} \) is the dc heat transfer rate, \( k/d \) is the ratio of thermal conductivity and thickness of the Kapton insulator, \( T_t \) and \( T_b \) are the top and bottom sensor temperatures. Upon differentiation and by assuming that the error sources are uncorrelated, Eq B.1 becomes
\[
\frac{\delta \dot{Q}}{\dot{Q}} = \sqrt{\left( \frac{\delta (\frac{k}{d})}{\frac{k}{d}} \right)^2 + \left( \frac{k/l}{\dot{Q}} \delta (T_I - T_b) \right)^2}
\]

Eq B.2

Typical values for \(\delta(k/d)/(k/d)\) are about 1\% [25]. The top and bottom sensor temperature mismatch, \(\delta(T_r - T_b)\) were obtained from the sensor temperatures averaged for the first 30 ms of the blowdown test. For most sensors in test T174, \(\delta(T_r - T_b)\) is less than 0.5\(^\circ\)C. This leads to an heat flux uncertainty of about 5\%.

The uncertainty in Nusselt numbers need to account for errors in gas temperature determination. Denoting gas and metal temperatures by \(T_{gas}\) and \(T_{wall}\), and using rotor axial chord \(L\) as the reference length scale, Nusselt number can be defined as

\[
Nu = \frac{\dot{Q} L}{k_{wall} (T_{gas} - T_{wall})}
\]

Eq B.3

The uncertainty in Nusselt number are related to gas and metal temperature uncertainties by the following expression

\[
\frac{\delta Nu}{Nu} = \sqrt{\left( \frac{\delta \dot{Q}}{\dot{Q}} \right)^2 + \left( \frac{\delta T_{gas}}{T_{gas} - T_{wall}} \right)^2 + \left( \frac{\delta T_{wall}}{T_{gas} - T_{wall}} \right)^2}
\]

Eq B.4

Heat flux uncertainty is evaluated using Eq B.2. The uncertainty levels of the gas temperature \(\delta T_{gas}\) is about 2\(^\circ\)K [52]. There were three RTD's mounted at the base of the instrumented rotor blades. Their indicated metal temperatures were compared among tests, yielding an uncertainty estimate for the wall temperature \(\delta T_{wall}\) at about 2\(^\circ\)C.
Summing up all the contributions in Eq B.4, the Nusselt number uncertainty for test T174 were typically at 5-10%.
Appendix C.
Transient Heat Conduction Model of the Matrix Heat Exchanger

The analysis shown here was carried out while the author was trying to master the "art" of pre-heating the upstream radial temperature distortion generator. The solutions obtained here are unique, and not found in standard heat conduction references, for example, that by Carslaw and Jaeger\textsuperscript{1}.

The electrical heater wires are threaded from the upstream side of the honeycomb heat exchanger to the downstream side, see Fig. 2.4. Therefore the heating can be treated as uniform in the axial direction. The energy added increases the temperature at the matrix center and heat diffuses away from the heater cable, producing spatial temperature variations. This process can be modeled as a transient two-dimensional heat conduction problem as shown in Fig. C.1.

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{honeycomb_matrix_diagram.png}
\caption{Two Dimensional Heat Conduction in the Honeycomb Matrix.}
\end{figure}

For pure radial temperature distortions, heating is uniform along circumferential direction. The conduction process can be modeled as 1D. For discrete heating as shown in Fig. 2.4, sector A, diffusion occurs in both radial and circumferential directions, then the modeling has to be in 2D. In both cases the heat conduction process is unsteady.

### C.1. One Dimensional Transient Heat Conduction

To avoid the use of Bessel functions and simplify the algebraic manipulation, the annular shape of the wall is ignored and a simple one dimensional heat conduction model is used as in Fig. C.2. The curvature effects are small if the inner and outer radius ratio is close to 1. In this case the ratio is 0.7.

![1D Heat Conduction model](image)

The unsteady heat conduction equation is

\[
\frac{\partial^2 T}{\partial y^2} + \frac{W_i(y,t)}{k} = \frac{1}{\alpha} \frac{\partial T}{\partial t}
\]

Eq C.1

where \( T \) is temperature, \( t \) is time, \( k \) and \( \alpha \) are the thermal conductivity and thermal diffusivity, \( W_i(y,t) \) is the heat added. The heat source can be treated as a singular point, i.e. a delta function, \( W_i(y,t) = W(t)\delta(y) \). Let \( \theta = T - T_{\infty} \) , \( y' = y/b \), and omitting the prime ' , Eq
C.1 becomes

\[ \frac{\partial^2 \theta}{\partial y^2} + C(t) \delta(y) = \tau \frac{\partial \theta}{\partial t} \]

with \( C(t) = \frac{b^2 W(t)}{k} \)
\( \tau = \frac{b^2}{\alpha} \)

Eq C.2

The heat source function \( C(t) \) is related to the resistance of the heater wire and the electrical current applied. The boundary conditions at the wall are difficult to specify. The RTDFG casing temperature rises during the long heating time of several hours. Constant wall temperature \( \left( \partial \theta / \partial y \bigg|_{y = \pm b} = 0 \right) \) or no wall heat flux \( \left( \partial \theta / \partial y \bigg|_{y = \pm b} = 0 \right) \) is easier to treat mathematically, while the realistic wall conditions may lie in-between. For a constant and steady \( C(t) \), the solution to Eq C.2 for both boundary conditions are

1: \( \partial \theta / \partial y \bigg|_{y = \pm b} = 0 \) and \( \theta_{t=0} = 0 \)

\[ \theta(y, t) = C \sum_{n=0}^{\infty} \frac{1}{(n + \frac{1}{2})^2 \pi^2} \left( 1 - e^{-\frac{(n + \frac{1}{2})^2 \pi^2 t}{\tau}} \right) \cos(n + \frac{1}{2})\pi y \]

Eq C.3

2: \( \partial \theta / \partial y \bigg|_{y = \pm b} = 0 \) and \( \theta_{t=0} = 0 \)

\[ \theta(y, t) = C \left( \frac{t}{2\tau} + \sum_{n=1}^{\infty} \frac{1}{n^2 \pi^2} \left( 1 - e^{-n^2 \pi^2 \frac{t}{\tau}} \right) \cos n\pi y \right) \]

Eq C.4

The solutions with \( C = 1 \) are plotted in Fig. C.4. For low \( t/\tau \) the wall temperatures of the second solution do not depart from zero very much, and both solutions are nearly
identical. For longer time, the solution with constant wall temperature converges to a hat like steady state solution, energy input eventually balances that lost to the wall. For insulated wall, the temperature keeps rising due to energy conservation. The term $t/2\tau$ in Eq C.4 reflects this trend.

Eq C.3 was used to predict the honeycomb matrix behavior. During electrical heating, matrix temperatures at various radial positions were recorded and a least squares fit between data and Eq C.3 was performed. The model predictions are quite good, shown in Fig. C.5. The time constant $\tau$ was found to be about 7 hours, consistent with the fact that it takes 4-5 hours to prepare for a radial distortion test.

If the energy input varies with time, principles of superposition can be applied using Duhamel's superposition integral. If the solution shown in Eq C.3 or Eq C.4 is denoted as $\Theta(y, t)$, then the superposed solution is

$$\theta(y, t) = \int_0^t C(\xi) \frac{\partial \Theta(y, t - \xi)}{\partial t} d\xi$$

Eq C.5

If $C(t)$ varies in steps, integral Eq C.5 can be rewritten in sums through segment integration.

---

C.2. Two Dimensional Heat Conduction Model

Fig. C.3: Domain of 2D Heat Conduction Model.

Again a symmetrical rectangular coordinate is used to simulate an angular area for mathematical convenience. The dimensions of a and b, reflected in Fig 2.4, are 1.7" and 2". For a medium with nonisotropic thermal conductivity, the heat conduction equation can be written as

\[ k_x \frac{\partial^2 T}{\partial x^2} + k_y \frac{\partial^2 T}{\partial y^2} + W_i(x, y, t) = \rho c \frac{\partial T}{\partial t} \]

Eq C.6

where \( k_x \) and \( k_y \) are conductivity in tangential and radial directions, \( W_i(x, y, t) \) is the heat source, \( \rho c \) is the thermal capacity. Using nondimensional quantities \( \theta = T - T_{\text{ref}}, x' = x/a, y' = y/b, W_i(x, y, z) = W(t) \delta(x, y) \) and omitting \( ', \) Eq C.6 becomes

\[ \xi^2 \frac{\partial^2 \theta}{\partial x^2} + \frac{\partial^2 \theta}{\partial y^2} + C(t) \delta(x, y) = \tau \frac{\partial \theta}{\partial t} \]

Eq C.7
where $\zeta^2 = \frac{k_x}{k_y} \frac{b}{a^2}$

$C(t) = \frac{b^2}{k_y} W(t)$

$\tau = \frac{\rho C}{k_y} b^2$

Eq C.8

$\zeta^2$ is the ratio of thermoconductivity in radial and circumferential directions. Boundary conditions at $x = \pm 1$ is $\partial \theta / \partial x = 0$ due to symmetry. Solutions to Eq C.8 with constant wall temperatures or with zero heat flux at $y = \pm 1$ are

1: $\partial \theta / \partial x = 0, \partial \theta / \partial x = 0$

$$\theta(x, y, t) = C \sum_{m=0}^{\infty} \sum_{n=0}^{\infty} A_{m,n} \left( 1 - e^{-\lambda_{m,n}^2 t / \tau} \right) \cos(m \pi x) \cos((n + \frac{1}{2}) \pi y)$$

where $A_{0,n} = \frac{1}{2}$

$A_{m,n} = 1$

$$\lambda_{m,n}^2 = \zeta^2 m^2 \pi^2 + (n + \frac{1}{2})^2 \pi^2$$

Eq C.9

2: $\partial \theta / \partial y = 0, \partial \theta / \partial x = 0$

$$\theta(x, y, t) = C \left( \frac{t}{4\tau} + \sum_{m=0}^{\infty} \sum_{n=0}^{\infty} A_{m,n} \left( 1 - e^{-\lambda_{m,n}^2 t / \tau} \right) \cos(m \pi x) \cos(n \pi y) \right)$$

where $A_{0,0} = 0$

$A_{0,n} = A_{m,0} = \frac{1}{2}$

$A_{m,n} = 1 \quad$ for $m, n > 0$

$$\lambda_{m,n}^2 = \zeta^2 m^2 \pi^2 + n^2 \pi^2$$

Eq C.10

A least squares fit between Eq C.9 and the heating history data of sector A in is
shown in Fig. C.6 This fit indicates that the conductivity ratio in circumferential and radial direction is approximately 15 to 1, making it very difficult to sustain a circumferential temperature gradient. Solutions at two different times are shown in Fig. C.7. As can be seen, the temperature variations are predominantly in the radial direction.
Fig. C.4: 1D unsteady heat conduction model solutions.

Fig. C.5: Fit between data and 1D conduction model. Symbols are data taken from tip to hub at five locations.
Fig. C.6: Fit between data and 2D feat conduction model. Symbols are matrix temperatures at various radial and circumferential locations.

Fig. C.7: Model predicted temperature distribution in sector A of the matrix heat exchanger with different heating times.
Appendix D.

A Note on Time Averaging of Flow Variables

It was noted in Chapter 6 that the time-averaged total temperature was determined by averaging the instantaneous gas temperature through a complete flow cycle (hot streak passing period). Instantaneous temperature is calculated from flow variables such as density $\rho$, pressure $P$ and velocity $V$, using the following equation

$$T_T(t) = \frac{P(t)}{\rho(t)R} + \frac{V(t)^2}{2C_p}$$

Eq D.1

Because of the nonlinear nature of Eq D.1, the time-averaged total temperature generally differs from the value calculated from Eq D.1, based on time-averaged pressure, density and velocity. This difference can be very large, about 4% in the time-averaged temperature with a hot streak temperature ratio of 1.8. Most of the error is caused by the inappropriate use of the state equation, i.e. by assuming that $\overline{T} = \frac{\overline{P}}{\overline{\rho}R}$.

An order of magnitude estimate can demonstrate that the error is attributed to the large magnitude variations in density. Let us assume a notch-type temperature variation with constant static pressure. Specifically, let
\[
T(t) = \begin{cases} 
(1+\alpha)T_0 & 0 < t < \delta \\
T_0 & \delta < t < 1
\end{cases}
\]

and \[
\rho(t) = \begin{cases} 
(1+\alpha)^{-1}\rho_0 & 0 < t < \delta \\
\rho_0 & \delta < t < 1
\end{cases}
\]

with both \(\delta\) and \(\alpha\) much less than one. The time-averaged temperature and density are

\[
\bar{T} = (1 + \delta \alpha)T_0 \\
\bar{\rho} = (1 + \delta((1 + \alpha)^{-1} - 1))\rho_0 = (1 - \delta \alpha(1 - \alpha + O(\alpha^2)))\rho_0
\]

The "averaged temperature" based on \(\bar{\rho}\) is

\[
\bar{T}^e = T_0\frac{1}{1 - \delta \alpha(1 - \alpha + O(\alpha^2))} - 1 = T_0(1 + \delta \alpha - \delta \alpha^2 + \text{higher order terms})
\]

The error is \(\frac{\bar{T} - \bar{T}^e}{T_0} = \delta \alpha^2\). For values of \(\delta = 1/7\) and \(\alpha = 0.4\), the error is about 2%.

The inequality of \(\bar{P} \neq R\bar{\rho}\bar{T}\) can also be shown by examining the ratio

\[
\zeta = \frac{\bar{P}}{R\bar{\rho}\bar{T}} = 1 - \delta \alpha^2
\]

For large enough values of \(\delta\) or \(\alpha\), the ratio \(\zeta\) clearly deviates from unity.