Improved Performance and Durability in Gas Turbines Through Hot streak Management

A thesis submitted to the
Division of Research and Advanced Studies
of the University of Cincinnati
in partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE

in the Department of Aerospace Engineering and Engineering Mechanics
of the College of Engineering

2001

by
Vipul K. Gupta
B.Tech., Indian Institute of Technology, Kanpur, India 1998

Committee Chair: Dr. Paul D. Orkwis
Abstract

The temperature field exiting gas turbine combustors is highly non-uniform due to streaks of hot fluid directly downstream of combustor fuel nozzles. Hot streaks have been shown to limit the life of turbine blades. Modifying the combustor fuel nozzles can change the shape of these temperature non-uniformities or hot streaks. Different shapes can have different effect on blade surface temperatures. Adjusting the positions of hot streaks with respect to inlet guide vanes, known as “hot streak clocking”, can also be used to help control blade temperatures in gas turbines. It is common practice to modify blade counts to decrease CPU time requirements in turbine simulations used in design process, but the effect of this procedure on the results of the simulations has not been investigated. In the current investigation, the linearized unsteady deterministic source term approach is used to model hot streak migration. The basic idea is to create a source term that represents the effects of the unsteady temperature segregation phenomena in a steady state calculation. Effects of hot streak clocking, hot streak shape and blade count on the performance and surface temperatures of a model gas turbine are studied. It is shown that hot streak clocking, blade count and hot streak shape significantly affect blade surface temperature.
Acknowledgements

The author would like to thank Dr. Brian Mitchell and Dr. Scott Hunter for giving access to the Tacoma code & their guidance, and to Shane Lennon and Dave Topp for their help. A very special thanks to Robert Ogden, system administrator, Gas Turbine Simulation Lab., without his help computing these results in time would be an impossible task. Thanks are also in order for Dr. Mark Turner & Dr. San Mou Jeng for letting me use their computing resources when it was needed the most. I would also like to thanks Matthew Goettke for helping to monitor the simulations during my absence. I would like to express my deepest gratitude to my advisor Dr. Paul D. Orkwis for his guidance, encouragement, for sharing his experience with me and for the work done by him at GE AE. A very special mention to all my colleagues in GTSL, Girish Bhandari, Ashutosh Gangwar, Bojan Lukovic, Dinesh Godavarty, Louise Merz and Dennis Yoder, their valuable suggestion have given me a better understanding of the project. Finally I would like to thank my parents Vinod and Usha Gupta and my brother Utkarsh Gupta, for their love, support and understanding.
# Table of Contents

Abstract ..................................................................................................................................... i

Acknowledgements .................................................................................................................. ii

Table of Contents ................................................................................................................... 1

List of Figures ......................................................................................................................... 3

Introduction ............................................................................................................................... 7

1.1 Hot Streaks ......................................................................................................................... 9

1.2 Hot Streak Migration ......................................................................................................... 9

1.2.1 Past Experimental Efforts ............................................................................................ 11

1.2.2 Computational Efforts .................................................................................................. 14

2 Computational Procedure .................................................................................................. 19

2.1 TACOMA ...................................................................................................................... 19

2.1.1 Linearized Analysis .................................................................................................... 19

2.1.2 Boundary Conditions ................................................................................................. 22

2.2 Lumped Deterministic Source Term Approach ............................................................ 24
List of Figures

Figure 1.1: Velocity triangles, Kerrebrock-Mikolajczak effect ......................................................... 11

Figure 1.2: Time averaged surface temperature contours for LSRR rotor, circular HS with temperature ratio 2.0, experiments done by Dorney, et al.16 ................................................................. 16

Figure 2.1: Locations of boundaries for a typical annular cascade. Spanwise cross-section .... 23

Figure 3.1: Schematic of UTRC’s LSRR .......................................................................................... 32

Figure 3.2: Spanwise (mid span) section of the computational grid .................................................. 33

Figure 3.3: Axial section of rotor computational grid ...................................................................... 34

Figure 4.1: Hot Streak Definition at the inlet of first stator passage (Temperature Contours) .... 37

Figure 4.2.1: Time Average temperature contours for Rotor computed using baseflow+DST, Circular HS at mid passage of IGV (not clocked), 21-21-21 blade count (Case 1) ..................... 38

Figure 4.2.2: Time Average temperature contours for Rotor computed using baseflow+DST, Circular HS at mid passage of IGV (not clocked), 21-28-21 blade count (Case 2) ..................... 39

Figure 4.2.3: Time Average temperature contours for Rotor computed using baseflow+DST, Elliptical HS at mid passage of IGV, 21-21-21 blade count (Case 5) ........................................... 39

Figure 4.3.1: Instantaneous static temperature contours at 40% span (base flow + linear perturbations), Circular HS at mid passage of IGV (not clocked), 21-21-21 blade count (Case1) ....................................................................................................................................................... 40

Figure 4.3.2: Instantaneous static temperature contours at 40% span (base flow + linear perturbations), Circular HS at mid passage of IGV (not clocked), 21-28-21 blade count (Case 2) ....................................................................................................................................................... 41
Figure 4.3.3: Instantaneous static temperature contours at 40% span (base flow + linear perturbations), Circular HS impinging IGV (clocked), 21-21-21 blade count (Case 3) .................. 41

Figure 4.3.3: Instantaneous static temperature contours at 40% span (base flow + linear perturbations), Elliptical HS at mid passage of IGV (not clocked), 21-21-21 blade count (Case 5) ....................................................................................................................................................... 42

Figure 4.4: Instantaneous static temperature contours (Cycle Time T/2) at various spanwise locations, Circular HS at mid passage of IGV, 21-28-21 blade count (Case 2) ......................... 43

Figure 4.5.1: Time average static temperature contours at various spanwise locations, Circular HS at mid passage of IGV, 21-21-21 blade count (Case 1) .......................................................... 46

Figure 4.5.2: Time average static temperature contours at various spanwise locations, Circular HS at mid passage of IGV, 21-28-21 blade count (Case 2) .......................................................... 47

Figure 4.5.3: Time average static temperature contours at various spanwise locations, Elliptical HS at mid passage of IGV, 21-21-21 blade count (Case 5) .......................................................... 48

Figure 4.5.4: Time average static temperature contours at various spanwise locations, Elliptical HS Impinging IGV (clocked), 21-21-21 blade count (Case 7) .......................................................... 49

Figure 4.5.4: Time average static temperature contours at various spanwise locations, Elliptical HS Impinging IGV (clocked), 21-28-21 blade count (Case 8) .......................................................... 50

Figure 4.6.1: Base flow & Energy source term at 50% span, Circular HS at mid passage of IGV (not clocked), 21-21-21 blade count (Case 1) ............................................................................................ 51

Figure 4.6.2: Base flow & Energy source term at 50% span, Circular HS at mid passage of IGV (not clocked), 21-28-21 blade count (Case 2) ............................................................................................ 51

Figure 4.6.3: Base flow & Energy source term at 50% span, Circular HS Impinging IGV (clocked), 21-21-21 blade count (Case 3) ............................................................................................ 52

Figure 4.6.4: Base flow & Energy source term at 50% span, Circular HS Impinging IGV
Figure 4.6.5: Base flow & Energy source term at 50% span, Elliptical HS at mid passage of IGV (not clocked), 21-21-21 blade count (Case 5) ..................................................................................... 53

Figure 4.6.6: Base flow & Energy source term at 50% span, Elliptical HS at mid passage of IGV (not clocked), 21-28-21 blade count (Case 6) ..................................................................................... 53

Figure 4.6.7: Base flow & Energy source term at 50% span, Elliptical HS Impinging IGV (clocked), 21-21-21 blade count (Case 7) ..................................................................................... 54

Figure 4.6.8: Base flow & Energy source term at 50% span, Elliptical HS Impinging IGV (clocked), 21-28-21 blade count (Case 8) ..................................................................................... 54

Figure 4.7.1: IGV surface temperature profile, Circular HS, 21-21-21 blade count (Case1-Mid Passage & Case3-Impinging) ........................................................................................................5 6

Figure 4.7.2: IGV surface temperature profile, Elliptical HS, 21-21-21 blade count (Case10-Mid Passage & Case6-Impinging) ........................................................................................................5 7

Figure 4.8: IGV surface temperature, Circular HS, Impinging IGV (Case3) …………………... 58

Figure 4.9.1: Time average rotor surface temperature profiles, Circular HS, 21-21-21 blade count (Case1-Mid Passage & Case3-Impinging) .................................................................................... 59

Figure 4.9.2: Time average rotor surface temperature profile, Circular HS, 21-28-21 blade count (Case2-Mid Passage & Case4-Impinging) .................................................................................... 60

Figure 4.9.3: Time average rotor surface temperature profile, Elliptical HS, 21-21-21 blade count (Case10-Mid Passage & Case7-Impinging) .................................................................................... 61

Figure 4.10.1: Time average rotor pressure surface temperature, Circular HS, 21-21-21 blade count .............................................................................................................................................. 63

Figure 4.10.2: Time average rotor suction surface temperature, Circular HS, 21-21-21 blade...
Figure 4.11.1: IGV pressure surface temperature, Impinging hot streak, 21-21-21 blade count...

Figure 4.11.2: Time average rotor pressure surface temperature, mid passage, 21-21-21 blade count...

Figure 4.12.1: IGV surface temperature profile, Impinging hot streak, 21-21-21 blade count (Case3-Circular HS & Case7-Elliptical HS)

Figure 4.12.2: Time average rotor surface temperature profile, Mid passage hot streak, 21-21-21 blade count (Case9-Circular HS & Case10-Elliptical HS)

Figure 4.13.1: Time average rotor surface temperature profile, Circular HS at mid passage (Case1-blade count 21-21-21 & Case2-blade count 21-28-21)

Figure 4.13.2: Time average rotor surface temperature profile, Elliptical HS at mid passage (Case5-blade count 21-21-21 & Case6-blade count 21-28-21)

Figure 4.14.1: Exit tangential flow angle, Inlet guide vane, Elliptical HS at mid passage

Figure 4.14.2: Time average rotor surface temperature profile, Elliptical HS at mid passage, 21-21-21 blade count (Case 10 HS temp. ratio=1.5 & Case 5 HS temp. ratio=2.0)

Figure 4.15: Mid span pressure profiles for IGV & Rotor

Figure 4.16: Exit tangential flow angle

Figure 4.17.1: Time averaged rotor surface temperature, circular HS at mid passage with 21-21-21 blade count

Figure 4.17.2: Time average rotor surface temperature profile, Circular HS at mid passage, 21-21-21 blade count (Case 1 poor grid & Case 10 fine grid)
Modern gas turbine engines can be designed with high turbine inlet temperature because of the availability of improved alloys and efficient cooling arrangements. The drive toward low fuel consumption and maximum gas turbine efficiency for commercial aircraft have also led to increases in combustor exit temperature, which in turn has had a direct impact on the durability of first-stage turbine blades. In order to keep turbine blade wall temperatures below critical levels, complex internal cooling schemes are designed to lay a film of relatively cool air adjacent to the blade. During the design of the first-stage turbine, flow field analyses that assume a uniform turbine inlet temperature profile are used to predict the airfoil surface temperature distributions. These wall temperature distributions are subsequently used to guide the design of the internal cooling schemes. One of the flow phenomena absent in many (or typical design) analyses is the spatial and temporal gas temperature non-uniformity generated by the combustor. These temperature distortions add complexity to turbine flows, complicating the heat transfer processes. The temperature non-uniformity in flow exiting from combustor results in interesting unsteady phenomena that needs to be accounted.

Typical mixing plane or steady multi-stage analyses do not simulate the effect of this unsteady phenomenon because they smear the circumferential variations and thereby obliterate the basis for the effect. Unsteady calculations are the only apparent hope for including the effect of temperature non-uniformities in the solution. Unsteady CFD codes are capable of simulating the phenomena but are generally too slow in reaching cyclic convergence to be of use in an overnight design time frame. Progress is rapidly being made to improve the
efficiency of these schemes, but their use in a day-to-day design sense is probably still a few years off.

What is needed at the present time is a way to include the effect of unsteadiness in a steady solver, if even in an approximate sense. Deterministic source terms (DST) offer a possible approach to solving this problem. In this idea, a source term is inserted into the governing equations that mimic the effect of some unsteady phenomena in much the same way that the effect of turbulence (the ultimate unsteady phenomena) is incorporated via turbulence models in the Reynolds averaging approach.

The lumped deterministic stress (LDS) approach has been developed in the past few years to determine the exact form of the unsteady source\textsuperscript{1,2} term from fully unsteady solutions. The idea can be applied in a postdictive sense once the more detailed solution is known. A variation of this approach has been employed recently by Busby, et al.\textsuperscript{2}, who used a lower fidelity (inviscid) set of equations to compute approximate source terms in a relatively quick time.

An even more aggressive approach used by Orkwis, et al.\textsuperscript{3}, was to utilize linear unsteady techniques to determine the source terms. These techniques have been used extensively for aeromechanics problem and can produce reasonably approximate unsteady flow fields in a fraction of the computer time as compared to nonlinear analyses. The difficulty with the approach is that the perturbations from the steady state solution are linear in nature; hence, they do not include the full unsteady flow physics.

Although linear, these techniques do offer a reasonable approximation to the unsteady
perturbation flow field. In the current study these techniques are used to compute the DST in an approximate way.

1.1 Hot Streaks

In gas turbine engine flows exiting the combustor can contain large circumferential and radial gradients in temperature, which results in hot spots, and when these hot spots migrate through the first-stage turbine they result in Hot Streaks. These temperature gradients arise from the combination of combustor core flow with combustor bypass and combustor surface cooling flows. It has been shown both experimentally and numerically\textsuperscript{4,5} that temperature gradients, in absence of total pressure non-uniformities, do not alter the flow within the first-stage turbine stator but do have a significant impact on the secondary flow and wall temperature of the first-stage rotor. Combustor hot streaks, which can typically have stagnation temperatures twice the free stream stagnation temperature, increase the extent of the secondary flow in the first-stage rotor and significantly alter the rotor surface temperature distribution compared to one without hot streaks.

1.2 Hot Streak Migration

The presence of hot streaks has taken on greater importance to turbine designers as the drive toward increased engine efficiency has resulted in higher combustor exit gas temperatures. In fact, the combustor exit flow may contain regions where the temperature exceeds the allowable metal temperature by 260-520 °C\textsuperscript{4}. Understanding the secondary flow and heat transfer effects due to the combustor hot streaks is essential for increasing the
performance and durability of high-pressure turbines, as well as optimizing internal and film cooling schemes. As a point of reference, it has been estimated that an error in predicting the time-averaged temperature on a turbine rotor of 55 °C can result in an order of magnitude change in blade life\textsuperscript{6,7}.

 Preferential surface heating is the name sometimes used to describe the unsteady phenomena of rotor temperature segregation caused by combustor hot streaks. Another name associated with hot streak surface heating is the Kerrebrock-Mikolajczak\textsuperscript{8} effect. These researchers were the first to describe why unsteady effects associated with compressors lead to increased pressure side heating. This effect is even more pronounced in turbines due to larger circumferential temperature variations. The description of the effect is based on the fact that the hot streak is moving at speeds significantly different from the surrounding fluid. The velocity diagram shown in Figure 1.1 is helpful in explaining the heating process. For a given inlet total pressure, streaks, whether hot or cold, have the same flow direction and are accelerated to the same Mach number as the surrounding fluid while passing through the stator (Munk and Prim).\textsuperscript{28} Because of the density differences, the hot stream is traveling faster and the cold stream is traveling slower than the surrounding fluid. \((C_H > C_F > C_C)\). As shown in Figure 1.1, the hot and cold fluid will be parallel to the free stream fluid in the stationery frame of reference, i.e., \(\nabla_H = \nabla_F = \nabla_C\). In the rotating frame of reference, however, they will not only have different magnitudes \((W_H > W_F > W_C)\) but different directions \((\vec{A}_C > \vec{A}_F > \vec{A}_H)\). The different magnitudes of the velocities will generally cause the hot streak to have a positive incidence and a higher relative total pressure than the surrounding fluid, both of which cause it to be driven toward the rotor pressure surface. Similarly, cold streaks and cooled stator wakes generally have a negative
incidence and a lower relative total pressure than the surrounding fluid, which cause them to be driven toward the rotor suction surface.

Figure 1.1: Velocity triangles, Kerrebrock-Mikolajczak effect

1.2.1 Past Experimental Efforts

The turbine configuration for which the most experimental hot streak data exists is the Large Scale Rotating Rig (LSRR) located at the United Technologies Research Center. The LSRR is a large-scale, low speed, rotating wind tunnel facility designed to simulate the flow field in axial-flow turbomachinery. For the hot streak experiments, the LSRR was configured to
resemble the first stage of a high-pressure turbine, typical of those used in modern commercial jet engines. The turbine stage consisted of 22 stator airfoils and 28 rotor airfoils.

In the first experimental study one hot streak was introduced through a circular pipe at 40% span, midway between two stator airfoils of the LSRR\textsuperscript{4}. The temperature of the hot streak was twice that of the surrounding inlet flow, whereas the static and stagnation pressure were identical to the free stream. The hot streak was seeded with CO\textsubscript{2} and the path of the hot streak was determined by measuring CO\textsubscript{2} concentrations at various locations within the turbine stage. A second experimental study\textsuperscript{9} was conducted using the same configuration as in the first experiment, except that the temperature of the hot streak was only 1.2 times that of the surrounding inlet flow and the flow coefficient was slightly increased.

Butler, et al. observed significant changes in the rotor surface flow patterns when the hot streak was present\textsuperscript{4}. Without the hot streak the maximum normalized CO\textsubscript{2} concentration on the rotor pressure surface was approximately 2.0, which was equal to the maximum circumferentially averaged level at the rotor inlet. The maximum CO\textsubscript{2} level detected on the rotor pressure surface was approximately 2.6 when the hot streak was introduced, which was much higher than the maximum circumferentially averaged inlet value of 2.1. In addition, the hot streak generated a noticeable increase in the span wise migration of the flow on the pressure surface of the rotor.

As observed by Butler, et al., the presence of the hot streak results in a segregation of the hot and cold gases. The hot streak fluid migrates towards the pressure surface of the rotor and spreads radially over the surface; the colder fluid migrates towards the suction surface of the rotor and is squeezed towards midspan by the secondary flows. The separation of the
hot and cold gases in the rotor can be partially accounted by the difference in relative inlet flow angles between the hot and cold gas streams, i.e., the Kerrebrock-Mikolajczak effect. As explained by Butler, the experimental results indicated that the hot streak does not affect the stator streamline pattern if the total pressure distribution is matched to that of the surrounding free stream. This leads to a condition where the static pressure, total pressure and absolute flow angle of the fluid inside and outside the hot streak are the same at the exit of the stator. The difference in the total temperature (and density) inside and outside the hot streak leads to a difference in the absolute velocity. Transforming the velocity to the relative frame results in a difference between the rotor relative inlet flow angle and the velocity magnitude. Thus, the hot fluid stream is oriented more toward the rotor pressure surface and moves at a greater relative speed.

As noted by Butler, et al., the hot streak also produces radial and circumferential gradients in the rotor relative inlet conditions. These experimental observations were corroborated by the inviscid theory of Lakshminarayana.

A third experimental study was performed using the LSRR to study the effects of hot streak shape on the migration pattern and to provide a data set that could be used to validate two-dimensional simulations. In this experiment, one hot streak was introduced in the form of a two-dimensional jet from the hub to the tip between two stator airfoils of the LSRR. In this experiment the stator vanes were rotated (closed) down 7 degrees with respect to the tangential direction relative to the design point operating conditions.
1.2.2 Computational Efforts

The first numerical simulation of hot streak migration in the LSRR was performed by Rai and Dring\textsuperscript{5} using a two-dimensional Navier-Stokes analysis. The simulation was significant because it roughly captured the migration patterns of the hot streak through the turbine. The predicted time-averaged rotor surface temperatures, however, did not show close agreement with the experimental data. The predicted suction surface temperatures were greater than the experimental values, while the pressure surface temperatures were significantly lower than the experimental data indicated. Originally it was thought that the discrepancies between the numerical and experimental data were caused by the use of a hot streak temperature ratio of 1.2 in the simulations, whereas a value of 2.0 was used in the experiments. Krouthen and Giles\textsuperscript{12} performed additional two-dimensional flow simulations using a hot streak temperature ratio of 2.0. In addition, they studied the influence of transition and flow coefficient on the rotor surface heating. Despite a more accurate representation of the experimental hot streak, the predicted results of Krouthen and Giles did not show significant improvement over the solutions presented by Rai and Dring. These simulations showed that three-dimensional simulation techniques are necessary to accurately capture the rotor pressure surface heating observed in the experiments.

In order to determine if a two-dimensional simulation could yield accurate results for a two-dimensional hot streak, Sharma, et al. performed experiments in which they replaced the circular hot streak with a two-dimensional hot streak which extended from the hub to the tip of the LSRR\textsuperscript{11}. When two-dimensional simulations were performed using the hot streak, the predicted results showed good agreement with the experimental data\textsuperscript{13,14}. These results showed that two-dimensional flow simulations can provide accurate information regarding the
physics of hot streak migration, provided the hot streak has a constant shape and extends from the hub to the tip. To accurately model circular, or other three-dimensional, hot streak shapes a three-dimensional flow analysis is required.

The simulations performed by Dorney, et al., resulted in several other interesting observations concerning comparisons between the numerical and experimental data \(^{13,14}\). In the hot streak experiments, the \(\text{CO}_2\) concentrations (used to track the hot streak) along the surface of the rotor were determined by drawing samples of the gas through the static pressure taps. The suction force may have caused \(\text{CO}_2\) gas from well above the airfoil surface to be included in the samples. Integrating the temperature through the boundary layer in the numerical simulations tested this hypothesis. The resulting temperature distribution showed excellent agreement with the experimental data, and collapsed the temperature distributions for different hot streak temperature ratios (1.5, 2.0 and 2.5) onto similar curves.

The first three-dimensional simulations of hot streak migration in the LSRR were performed by Dorney, et al.\(^{15,16}\), and by Takahashi and Ni\(^{17,18}\). These simulations illustrated that accurate modeling of the hot streak geometry leads to the significant rotor pressure surface heating observed in the experiments. As a part of the three-dimensional research studies, animations were created of the turbine flow field. The animations showed that the hot streak is relatively unaffected, as it is convected through the stator passage, except that the width of the hot streak decreases due to flow acceleration. As the hot streak migrates into the rotor passage it is chopped by the passing rotor blades. The hot streak assumes a V-shape as it moves through the rotor passage, and is convected downstream as discrete temperature eddies. The simulation of Takahashi and Ni\(^{18}\) suggested that the temperature segregation process does not
occur in second-stage stator passages, although later simulations by Dorney, et al.\textsuperscript{19}, showed that temperature segregation effects are present in second-stage blade rows for certain airfoil-count ratios. The work of Takahashi and Ni\textsuperscript{17} verified that unsteady flow simulations are needed to capture the correct physics of hot streak migration. This fact is significant since most modern design systems still rely heavily on two- and three-dimensional steady flow analyses.

![Figure 1.2: Time averaged surface temperature contours for LSRR rotor, circular HS with temperature ratio 2.0, experiments done by Dorney, et al.\textsuperscript{16}](image)

Given the fact that combustor hot streaks exist in gas turbine engines, designers have attempted to devise methods for alleviating their adverse effects. Internal and film cooling schemes have been used for years to protect the surfaces of airfoils. Only recently, however, have engineers had the ability to optimize the cooling schemes through the use of numerical simulations. A study by Dorney and Davis\textsuperscript{16} showed that three-dimensional flow analyses can be used to strategically place film cooling holes on the surface of the rotor to eliminate the harmful effects of hot streaks. Perhaps more important has been the advent of passive techniques for
controlling the effects of hot streaks. The numerical works of Dorney and Gundy-Burlet\textsuperscript{20,21}, and Takahashi, et al.\textsuperscript{22}, along with the experimental work of Roback and Dring\textsuperscript{9} have shown that first-stage turbine stators can be “indexed” or “clocked” with respect to the hot streaks to minimize airfoil heating. Recall that in the original experiments of Butler, et al. the hot streak was introduced between two adjacent first-stage stator airfoils. This allows the hot streak to travel undisturbed through the stator passage, until the hot gases are segregated in the rotor passage. If the first-stage stators are aligned with the hot streaks, however, then the hot streak is forced to impact the stator and convect along its suction and pressure surfaces. This introduces a heavy thermal load onto the stator airfoils, but these airfoils must be designed to accept stoichiometric temperatures anyway\textsuperscript{20,21}. The advantages of this concept are two-fold. First, the hot streak is forced to mix with the cooler fluid of the stator wake. This acts to dissipate the hot streak and reduce its maximum temperature. Second, the hot streak now has a tendency to move toward the suction surface of the rotor along with the stator wake, as colder stator wakes move toward the suction surface of the rotor, the Kerrebrock-Mikolajczak\textsuperscript{8} effect. The dissipated hot streak is rapidly convected along the suction side of the rotor passage before it can do any damage. By the time the hot streak reaches the second stage of the turbine it has been sufficiently weakened such that it does not cause any damage. Gundy-Burlet and Dorney\textsuperscript{23} have recently performed a parametric study to evaluate the effects of radial placement of the hot streak at the inlet of the turbine. The results of this study have shown that placing the hot streak where it can be entrained in the secondary flows has a significant impact on the integrated heat transfer.

Most of the LSRR simulations described to this point were performed using adiabatic surface conditions. In an actual operating environment, however, there can be significant
amounts of heat transfer. A recent study by Gundy-Burlet and Dorney\textsuperscript{24} showed that three-dimensional hot streak simulations including heat transfer effects can be used to locate additional regions of the turbine which are susceptible to burning (e.g., the hub, the aft 50\% of the rotor suction surface near the tip, etc.).

The development of approximate techniques for incorporating hot streak effects has recently been undertaken by Busby, et al. and Orkwis, et al.\textsuperscript{2,3}. Both approaches utilize the lumped deterministic stress (LDS) approach applied previously by Sondak, et al.\textsuperscript{1} to incorporate the unsteady effects of temperature segregation into steady state computations. The first group employed nonlinear unsteady inviscid simulations to compute the LDS terms and the second employed a linear unsteady inviscid approach. Computation time for both was reduced considerably as compared to nonlinear unsteady viscous calculations, and both represent convenient approaches for application in the design environment. However, the extent to which they can be accurately employed has yet to be verified.

Some of the unresolved issues in hot streak migration studies include: 1) the influence of the hot streak/airfoil count ratio, 2) the effects of hot streak shape on airfoil heating, 3) the effects of hot streak unsteadiness and 4) the degree to which approximate techniques can model the temperature segregation flow physics. These issues are addressed in this research study.
2 Computational Procedure

2.1 TACOMA

The computational code used in this study is a General Electric Aircraft Engines proprietary code known as TACOMA or Turbine And COMpressor Analysis. TACOMA is 3D structured non-linear and linear Euler/Navier-Stokes equation solver for turbomachinery blade rows. It is a cell centered explicit solver and uses a Runge-Kutta time marching scheme with local time steps, multigrid and residual averaging. Either the inviscid (Euler), laminar or Reynolds averaged form of the Navier-Stokes equations can be solved. The Reynolds averaged form of the Navier-Stokes equations are closed using k-\(\overline{\text{i}}\) two equation turbulence models. The equations are cast in a rotating reference frame so either rotating or stationary blade rows can be computed. With the use of source terms, adjacent blade rows of a multi-blade machine can be simulated, as can cooling flows. The code is set up to solve on multi-block grids.

Linear unsteady solutions may be computed for forced response by reading “gust” data. It may be obtained using 1D or 2D (Giles) “non-reflecting” boundary conditions.

2.1.1 Linearized Analysis

In the linearized approach the temporal unsteadiness is represented by a Fourier series description, which can be written in the exponential form of a complex valued function. When this is done the time derivative is removed from the system. Then, instead of solving for an unsteady time history of a real valued function, one solves for the steady state of
a complex valued function. In addition, perturbation products are ignored, so that the perturbations themselves become functions of a linear operator.

The unsteadiness is passed between blade rows by transforming the spatial non-uniformities in the circumferential direction into their corresponding temporal variations in the rotating reference frame. A Fourier series is then performed to determine the appropriate discrete frequencies that exist in the solution. A linear solution is solved for each existing frequency and the full unsteady solution is developed by superposition once the solution associated with each frequency is obtained. The discrete frequency solutions can be obtained in an inherently parallel manner because of the linear nature of the perturbations.

For a general derivation of the linearized approach equation, consider 2-D unsteady Euler equation (for simplicity) as found in classic CFD text by Anderson, et al.\textsuperscript{25}

\[
\frac{\partial \tilde{Q}}{\partial t} + \frac{\partial \tilde{F}}{\partial x} + \frac{\partial \tilde{G}}{\partial y} = 0
\]

(2.1)

where \( \tilde{Q} \) is the vector of conservative variables, \( \tilde{F} \) and \( \tilde{G} \) are the flux vectors.

Now, decomposing the unsteady flow field variables into mean and perturbed quantities, as represented by the first-order perturbation series

\[
\tilde{Q} = \bar{Q}(x, y) + q'(x, y, t) = \bar{Q}(x, y) + \hat{q}(x, y) e^{j\omega t}
\]

\[
\tilde{F} = \bar{F}(x, y) + f'(x, y, t) = \bar{F}(x, y) + \hat{f}(x, y) e^{j\omega t}
\]

(2.2)

\[
\tilde{G} = \bar{G}(x, y) + g'(x, y, t) = \bar{G}(x, y) + \hat{g}(x, y) e^{j\omega t}
\]
where \( \bar{Q}, \bar{F} \) and \( \bar{G} \) are the vectors of mean or steady flow conservation variables, \( q', f' \), \( g' \) are the vectors of first-order perturbations in conservation variables, \( \hat{q}, \hat{f} \) and \( \hat{g} \) are the complex amplitudes of the first-order perturbation in conservative variables and \( \omega \) is the frequency of excitation. After substituting Eq. 2.2 in Eq. 2.1 and simplifying, we can get mean flow Euler equation as,

\[
\frac{\partial \bar{F}}{\partial x} + \frac{\partial \bar{G}}{\partial y} = 0
\]  

(2.3)

and the linearized equation in the frequency domain is,

\[
j\omega \hat{q}(x, y) + \frac{\partial \hat{f}}{\partial x} + \frac{\partial \hat{g}}{\partial y} = 0
\]  

(2.4)

Note that the \( \hat{q}(x, y) \) is a complex valued function which can be further resolved into

\[
\hat{q} = \hat{q}_r + i\hat{q}_i
\]

and using Eq. 2.4 both the real and imaginary part of \( \hat{q}(x, y) \) can be computed. Finally, the unsteady solution can be computed by adding the real part of the linearized solution to the mean or steady flow solution.
\[
\tilde{q} = \sqrt{q_R^2 + q_I^2}
\]
\[
\theta = \tan^{-1} \frac{q_I}{q_R}
\]

(2.5)

\[
\tilde{Q} = \bar{Q} + \text{Re}(\tilde{q}e^{i(\theta + \omega \tau)})
\]

\[
\tilde{Q} = \bar{Q} + \tilde{q} \cos(\theta + \omega \tau)
\]

where \( \tilde{q} \) is the magnitude and \( \theta \) the phase of the perturbation, and \( \tilde{Q} \) is the unsteady solution at time \( t \).

2.1.2 Boundary Conditions

Periodic boundary condition – because the linearized Euler equations are linear, any arbitrary blade motion with frequency \( \bar{\Omega} \) may be thought of as composed of a sum of traveling wave modes, each mode having a fixed interblade phase angle, \( \bar{\epsilon} \). Each of these modes may be analyzed individually then summed to determine the total unsteady flow solution. Hence, without loss of generality, we assume that the unsteady flow has a fixed interblade phase angle, \( \bar{\epsilon} \), from blade passage to blade passage. The flow field may then be computed in a single blade passage by applying complex periodicity conditions along the upstream and downstream periodic boundaries. For annular cascades, the unsteady complex periodicity condition takes the form
Figure 2.1: Locations of boundaries for a typical annular cascade. Spanwise cross-section

\[ Q'(\xi, \theta + \Theta_G, r) = TQ'(\xi, \theta, r)e^{i\sigma} \]  \hspace{1cm} (2.6)

where \( \Theta_G \) is the angular distance between adjacent blades, and \( T \) is a matrix which rotates the velocity vector in the \((\eta, \zeta)\) plane through the angle \( \Theta_G \), and the interblade phase angle is defined as

\[ \sigma = -(360) * \frac{\text{No. of inlet guide vanes}}{\text{No. of rotor blades}} \text{ (deg)} \]  \hspace{1cm} (2.7)
Far-field boundary condition – as the computational domain must be finite in extent in the axial direction, non-reflecting boundary conditions are required at the upstream and downstream far-field boundaries to prevent spurious reflections of outgoing waves. In this study 1-D (Giles) non-reflecting boundary conditions are used. The inlet gust information for every mode is computed by Fourier decomposition of solution variables. The constant part of the Fourier decomposition is used as the boundary condition for the mean or steady flow and the perturbations are used as the boundary conditions for the linearized analysis.

2.2 Lumped Deterministic Source Term Approach

Deterministic source terms (DST) offer a means to include the effects of solution unsteadiness in time mean computations. They can be derived exactly in a postdictive manner once an unsteady solution has been obtained. Predictive models of the source terms can be generated once the general behavior of the unsteady effects has been characterized. The easiest way to understand how the DST can be obtained is to follow their general derivation, as described next.

Consider the general form of the unsteady energy equation (written in 2D) for inviscid, non-heat conducting flow (for simplicity) as found in CFD text by Anderson, et al.²⁵

\[
\frac{\partial E_t}{\partial t} + \frac{\partial [(E_t + p)u]}{\partial x} + \frac{\partial [(E_t + p)v]}{\partial y} = 0
\]

(2.8)

where \( E_t \) is the total energy per unit volume, \( p \) is the pressure, and \( u \) and \( v \) are velocity components. (Note that Equation (2.8) was written for the flow in an absolute reference frame to
simplify the explanation of the approach. The energy equation in the rotor relative frame is modified slightly to account for relative work terms. See Lyman\textsuperscript{26} for an explanation of energy equations forms that account for rotation.)

Nonlinear unsteady solution techniques obtain the solution in time from Equation (2.8) by moving the spatial derivatives to the right hand side (RHS) and discretizing, i.e.,

$$\frac{\partial E_t}{\partial t} = \left\{ \frac{\partial [(E_t + p)u]}{\partial x} + \frac{\partial [(E_t + p)v]}{\partial y} \right\}$$ \hspace{1cm} (2.9)

The same equations are used to solve for steady state solutions, in which Equation (2.9) is used to relax the solution to a steady state, i.e., left hand side (LHS) = 0. An equivalent way to describe this approach is to call the RHS of Equation (2.9) the negative of some residual, \(R(Q)\), and drive that residual to zero. This last description will take on greater meaning later.

Equation (2.9) is the governing equation for unsteady inviscid flows. It continues to govern the unsteady fluid mechanics even if the variables are decomposed into separate components. The typical DST decomposition begins by splitting variables into their deterministic and stochastic components, i.e., \(q = \bar{q} + q'\), where \(\bar{q}\) is the deterministic portion and \(q'\) is the stochastic. The latter component is then modeled using some sort of turbulence model, whereas the former can be simulated by the technique. If the stochastic components are ignored (since they are represented by turbulence models and the test case is assumed inviscid) then the next step in the derivation is to split the deterministic quantity, \(\bar{q}\), into a mean value, \(\bar{\bar{q}}\), and a deterministic fluctuation, \(q''\). If these expressions are substituted into Equation (2.9) and the stochastic fluctuations are ignored the equation may be rewritten as
\[
\frac{\partial}{\partial t} (\overline{E_i} + E_i^*) = \frac{\partial}{\partial x} [(\overline{E_i} + E_i^*) + p + p^* (u + u^*)] \\
- \frac{\partial}{\partial y} [(\overline{E_i} + E_i^*) + p + p^* (v + v^*)]
\]
(2.10)

Equation (2.10) is simply a restatement of Equation (2.9). It can be further expanded into

\[
\frac{\partial}{\partial t} (\overline{E_i} + E_i^*) = \left\{ \frac{\partial}{\partial x} [(\overline{E_i} + p)u] + \frac{\partial}{\partial y} [(\overline{E_i} + p)v] \right\} \\
- \left\{ \frac{\partial}{\partial x} [(\overline{E_i} + p)u^*] + \frac{\partial}{\partial y} [(\overline{E_i} + p)v^*] \right\} \\
- \left\{ \frac{\partial}{\partial x} [(E_i^* + p^*)u] + \frac{\partial}{\partial y} [(E_i^* + p^*)v] \right\} \\
- \left\{ \frac{\partial}{\partial x} [(E_i^* + p^*)u^*] + \frac{\partial}{\partial y} [(E_i^* + p^*)v^*] \right\}
\]
(2.11)

The time average of the LHS and the second and third brackets on the RHS are identically zero by definition once Equation (2.11) is time averaged. Hence, the governing equation (to be solved) for the time average of a flow with unsteady perturbations is

\[
0 = -\left\{ \frac{\partial}{\partial x} [(\overline{E_i} + p)u] + \frac{\partial}{\partial y} [(\overline{E_i} + p)v] \right\} \\
- \left\{ \frac{\partial}{\partial x} [(E_i^* + p^*)u^*] + \frac{\partial}{\partial y} [(E_i^* + p^*)v^*] \right\}
\]
(2.12)

or
Bracket one in Equation (2.12) is the same residual solved for the steady state by relaxation methods (see, for example, Equation (2.9)). Note that this term is already a mean quantity, hence, the time average is redundant. Bracket two represents the source terms that must be added to the steady equations to include the effect of unsteadiness. This mirrors the approach taken to include the unsteady effects of turbulence in steady state solutions inherent in Reynolds averaged form of the governing equations.

It should be recognized immediately that the solutions obtained from Equations (2.9) and (2.12) will be different and will respectively give the steady state and time average solutions, provided the correct DST is applied in the latter. The problem then becomes determining the correct form of the DST.

At first glance it might appear that solutions from linear approaches could not possibly be used to determine the DST because it is the nonlinear interaction between unsteady perturbations that leads to effects like temperature segregation. That is, unsteady temperature fluctuations are carried by unsteady velocity perturbations for flows with hot streaks. Whereas in linear unsteady approaches the perturbations are carried by the base flow, not by the coupled perturbed variable field.

To better understand this, consider first how the specific form assumed for the unsteadiness leads to the governing equations for linear unsteady solvers. These techniques solve for each perturbation frequency separately by imposing a Fourier series representation
of the temporal solution and linearizing the resulting equations. Complex numbers in the exponential form represent the series. The Fourier series transforms the equations from a set of unsteady nonlinear real variable equations, to a set of steady state linear complex variable equations. Therein lies the advantage of these schemes: the unsteady solution for each discrete frequency is obtained in a multiple of the time required to obtain steady state results (typically about twice as long,) and the linear perturbation assumption allows each frequency to be solved in parallel. This results in a considerable time reduction as compared to that needed for a nonlinear unsteady approach.

Unfortunately the solutions obtained in this manner do not satisfy Equation (2.12) exactly, but satisfy an approximate set of complex perturbation equations. Perturbation products are themselves defined to be zero under the small perturbation assumption. Therefore, Equation (2.12) is not itself satisfied by a linear unsteady approach. This causes a significant setback for the DST approach because the first bracket in Equation (2.12) is already effectively zero (as it represents the base flow) and cannot be used to form the deterministic stresses.

However, the DST can be reconstructed from the linear perturbation solution even though the linear solution procedure assumes away the perturbations. This is possible if the perturbations are a reasonable approximation of the actual nonlinear solution. In that case, even though the perturbations are themselves linear, a superposition solution can be created from the base flow plus perturbation variables and Equation (2.12) can be satisfied approximately. The DST in Equation (2.12) is clearly nonlinear even if it is approximated by originally linear perturbations.

Therefore, the approach required for determining the DST from a linear solver utilizes the time average of the nonlinear steady state residuals obtained with the base flow plus
linear perturbation solutions. It is in effect only the order in which the calculations are performed that changes, i.e.,

*Nonlinear solutions*

Step 1 - Time average the unsteady solution variables.

Step 2 - Compute the steady state residual.

*Linear solutions*

Step 1 - Compute the steady state residual for each instantaneous base flow plus perturbation solution.

Step 2 - Time average the residuals.

It should be pointed out that the DST obtained from nonlinear solutions are exact (within truncation error and assuming cyclic convergence), whereas the DST obtained from linear solutions will only be approximate. The accuracy of the latter depends upon the validity of the linear perturbation solutions. However, Orkwis, et al.\(^3\) have already shown that the individual perturbation components are reasonably accurate and provide a good approximation of the DST.

The key issue to understand when viewing the DST contours is the meaning of their spatial distribution. Strictly speaking the values of these terms represent redistributions of conserved quantities, not additional mass, momentum or energy. It is important to note that these changes are then convected along the streamlines to the rest of the solution. So
that given a flow with locally higher temperature that returned later to a lower temperature, the DST field would reflect this by being positive as the temperature increased and then negative as it decreased. It is then the integrated effect, in a substantial derivative sense that becomes important.
3 Turbine Geometry

The three-dimensional turbine model is based on UTRC’s Large Scale Rotating Rig (LSRR) geometry. The LSRR is a 1-1/2 stage high-pressure turbine with a 27 inch midspan radius, 6 inch span and airfoil aspect ratios of approximately unity. The turbine hub and casing are at constant radii. The axial gap between the first-stage stator and rotor is approximately 15% of the rotor axial chord, while the gap between the rotor and the second stage stator is approximately 50% of the rotor axial chord. The rotor tip clearance in the experimental rig is approximately 1% of span. In the experiments, the inlet Mach number to the first stage stator was 0.07 (flow coefficient of $\bar{N} = 0.78$) and the inlet flow was assumed to be axial. The rotor rotational speed was 410 rpm. The free stream Reynolds number was 100,000/inch. The midspan pressure ratio was set to $P_4/P_{t1} = 0.9505$, using the time & circumferentially averaged pressure ratio for each blade row from the non-linear runs performed by Dr. Sondak at Boston University. A large experimental database exists for this turbine, including time-averaged pressures at several spanwise locations on each airfoil, traverse data behind each airfoil, and surface flow visualizations.

The experimental configuration has 22 airfoils in the first stage stator row and 28 airfoils in each of the rotor and second stage stator rows, for a total of 78 airfoils. In order to study the effects of hot streaks and airfoil clocking different blade count ratios were investigated. Two different blade count ratios studied are, 21-21-21 (1-1-1 analysis) and 21-28-21 (3-4-3 analysis), and in both configurations airfoil geometry are scaled appropriately.
3.1 Grid

In all the simulations H-type grids are used for a single passage for each blade row.

First stage stator passage: 33 circumferential, 45 spanwise and 129 axial planes

First stage rotor passage: 49 circumferential, 49 spanwise and 129 axial planes

Second stage stator passage: 49 circumferential, 49 spanwise and 129 axial planes

Hence, for the complete 1-1/2 stage a total of 811,023 grid points were used. Initially
rotor passage simulations were performed on a grid of 33 circumferential & 45 spanwise planes, which was not sufficient to compute the turbulent linear analysis. The aspect ratio of cells near the leading edge is also very critical for turbulent linear solver.

Figure 3.2: Spanwise (mid span) section of the computational grid
3.2 Cases of Study

Table 3.1 contains a listing of the cases that are used for various study, such as hot streak shape effects, hot streak clocking effects and blade count ratio effects. For hot streak clocking study, two different hot streak positions are computed, one hot streak at mid passage (no clocking) and other hot streak impinging at first stage stator leading edge (hot streak clocked). Two different blade counts are studied, 21-21-21 (1-1-1 analysis) and 21-28-21 (3-4-3 analysis), the 21-21-21 blade count was selected so that the total enthalpy due to introduction of hot streaks at the inlet guide vane should be same in both blade count cases. Two different hot streak shapes, circular and elliptical were used for the hot streak shape studies. In the case of the elliptical hot streak at mid passage, case 5 (21-21-21 blade count) & case 6 (21-28-21 blade count), by mistake the temperature ratio in the hot streak was kept as 2.0 instead of 1.5. For hot streak shape study a new case (case 10) is computed with elliptical hot streak at mid passage, 21-21-21 blade count with temperature ratio of 1.5 on finer grid.
After seeing the results of these cases and comparing them with non-linear results it was observed that the results in hub region doesn’t match that well, the reason is grids of poor $y^+$ in hub region are used in linearized method. So one more case was studied with better grid to study the grid resolution effect.

<table>
<thead>
<tr>
<th>Case</th>
<th>Blade Count</th>
<th>HS Shape</th>
<th>Clock HS</th>
<th>Grid ($y^+$)</th>
<th>Rows Completed</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>21-21-21</td>
<td>Circular</td>
<td>No</td>
<td>Coarse</td>
<td>Single Stage</td>
</tr>
<tr>
<td>2</td>
<td>21-28-21</td>
<td>Circular</td>
<td>No</td>
<td>Coarse</td>
<td>Single Stage</td>
</tr>
<tr>
<td>3</td>
<td>21-21-21</td>
<td>Circular</td>
<td>Yes</td>
<td>Coarse</td>
<td>Single Stage</td>
</tr>
<tr>
<td>4</td>
<td>21-28-21</td>
<td>Circular</td>
<td>Yes</td>
<td>Coarse</td>
<td>Single Stage</td>
</tr>
<tr>
<td>5*</td>
<td>21-21-21</td>
<td>Elliptical</td>
<td>No</td>
<td>Coarse</td>
<td>Single Stage</td>
</tr>
<tr>
<td>6*</td>
<td>21-28-21</td>
<td>Elliptical</td>
<td>No</td>
<td>Coarse</td>
<td>Single Stage</td>
</tr>
<tr>
<td>7</td>
<td>21-21-21</td>
<td>Elliptical</td>
<td>Yes</td>
<td>Coarse</td>
<td>Single Stage</td>
</tr>
<tr>
<td>8</td>
<td>21-28-21</td>
<td>Elliptical</td>
<td>Yes</td>
<td>Coarse</td>
<td>Single Stage</td>
</tr>
<tr>
<td>9</td>
<td>21-21-21</td>
<td>Circular</td>
<td>No</td>
<td>Fine</td>
<td>Single Stage</td>
</tr>
<tr>
<td>10</td>
<td>21-21-21</td>
<td>Elliptical</td>
<td>No</td>
<td>Fine</td>
<td>Single Stage</td>
</tr>
</tbody>
</table>

Table 3.1: Table of simulations

* Indicates cases ran with temperature ratio of 2.0 instead of 1.5.
4 Results

4.1 Hot Streaks

Hot streaks are introduced at 40% span with tanh profile and temperature ratio of 1.5 in all cases. Two different shapes and clocking positions (Figure 4.1) are considered. In both the shapes, average inlet total enthalpy has been kept same by keeping the area of the hot region same (Please note that in figures 4.1 the scale factors for different figures are different).

In case of the elliptical hot streak,

\[
\frac{\text{major axis (spanwise direction)}}{\text{minor axis (circumferential direction)}} = \frac{2}{1}
\]
Various cases were computed for different studies and all of them depict the same migration pattern of hot streaks, i.e.,

- Hot fluid migrates toward the pressure surface of the rotor.
- The Individual segment of hot streaks takes on V-shape while passing through rotor passage since the convection speeds are lower near the blade.
surfaces than at mid passage.

- The pressure surface of the rotor exhibits highly elevated temperatures compared to the suction surface.

- The hot streak introduced at mid passage is largely unaffected while passing through the first-stage stator (Inlet Guide Vane, IGV) passage.

Figure 4.2.1: Time Average temperature contours for Rotor computed using baseflow+DST, Circular HS at mid passage of IGV (not clocked), 21-21-21 blade count (Case 1)
Figure 4.2.2: Time Average temperature contours for Rotor computed using baseflow+DST, Circular HS at mid passage of IGV (not clocked), 21-28-21 blade count (Case 2)

Figure 4.2.3: Time Average temperature contours for Rotor computed using baseflow+DST, Elliptical HS at mid passage of IGV, 21-21-21 blade count (Case 5)

In Figures 4.2 hub to tip is bottom to top.
Figures 4.2 shows that the pressure surface of the rotor has elevated temperatures in certain region, these regions are known as “hot spots”. These elevated temperatures are due to the following reasons,

- The introduction of hot streaks between first-stage stators, which allows the hot streak to migrate towards the pressure surface of the rotor because of relative flow angle considerations, i.e., the Kerrebrock-Mikolajczak\textsuperscript{8} effect.

- The relatively low velocity on the pressure side of the passage, which allows the hot streak to attain a relatively large residence time on the surface.

It can also be seen that surface temperature depends upon blade count (Figure 4.2.2) as well as hot streak shape (Figure 4.2.3). These effects will be discussed in the later sections.

![Cycle Time T/4](image1)

![Cycle Time T/2](image2)

![Cycle Time 3T/4](image3)

Figure 4.3.1: Instantaneous static temperature contours at 40% span (base flow + linear perturbations), Circular HS at mid passage of IGV (not clocked), 21-21-21 blade count (Case1)
Figure 4.3.2: Instantaneous static temperature contours at 40% span (base flow + linear perturbations), Circular HS at mid passage of IGV (not clocked), 21-28-21 blade count (Case 2)

Figure 4.3.3: Instantaneous static temperature contours at 40% span (base flow + linear perturbations), Circular HS impinging IGV (clocked), 21-21-21 blade count (Case 3)
Figure 4.3.3: Instantaneous static temperature contours at 40% span (base flow + linear perturbations), Elliptical HS at mid passage of IGV (not clocked), 21-21-21 blade count (Case 5)
Figure 4.4: Instantaneous static temperature contours (Cycle Time T/2) at various spanwise locations, Circular HS at mid passage of IGV, 21-28-21 blade count (Case 2)
In the rotor passage the hot streaks migrate toward the pressure surface of the rotor, it can be seen from figures 4.3 & 4.4, which shows the instantaneous static temperature contours for various cases at several spanwise locations. The V-shape of the individual hot streak segments in the rotor passage can also be observed in Figures 4.3 & 4.4.

This migration of hot streaks toward the pressure surface can also be observed in the quasi-time average temperature contours of rotor (Figures 4.5) computed using base flow plus deterministic source terms (DST). Figures 4.5 show the time average temperature contours of the rotor for various cases at several spanwise locations. The region of hot fluid near to the pressure surface of rotor is clearly visible in almost all spanwise locations.

Figure 4.5.4 (for Case 7, Elliptical HS, Impinging on IGV, 21-21-21 blade count) & Figure 4.5.3 (for Case 5, Elliptical HS at mid passage of IGV, 21-21-21 blade count) shows the effect of hot streak clocking, the hot region in Figure 4.5.4 is almost absent and hence results in lower rotor pressure surface temperature.

Figure 4.5.1 (for Case 1, Circular HS at mid passage of IGV, 21-21-21 blade count) & Figure 4.5.2 (for Case 2, Circular HS at mid passage of IGV, 21-28-21 blade count) shows the effect of blade count ratio, in the former case, the hot region is more prominent compared to the latter one hence 21-28-21 or 3-4-3 case will result in lower rotor pressure surface temperature. This can be verified in Figure 4.2.1 & 4.2.2.

Figure 4.5.1 (for Case 1, Circular HS at mid passage of IGV, 21-21-21 blade count) & Figure 4.5.3 (for Case 5, Elliptical HS at mid passage of IGV, 21-21-21 blade count) shows the effect of hot streak shape, in the latter case, the hot region is more prominent compared to the
former one hence circular hot streaks result in lower rotor pressure surface temperature. This can be verified in Figure 4.2.1 & 4.2.3.

The effect of hot streak clocking, hot streak shape & blade count ratio effect will be studied in detail in later sections.
Figure 4.5.1: Time average static temperature contours at various spanwise locations, Circular HS at mid passage of IGV, 21-21-21 blade count (Case 1)
Figure 4.5.2: Time average static temperature contours at various spanwise locations, Circular HS at mid passage of IGV, 21-28-21 blade count (Case 2)
Figure 4.5.3: Time average static temperature contours at various spanwise locations, Elliptical HS at mid passage of IGV, 21-21-21 blade count (Case 5)
Figure 4.5.4: Time average static temperature contours at various spanwise locations, Elliptical HS Impinging IGV (clocked), 21-21-21 blade count (Case 7)
Figure 4.5.4: Time average static temperature contours at various spanwise locations, Elliptical HS Impinging IGV (clocked), 21-28-21 blade count (Case 8)
Temperature contours of rotor base flow

Energy ($E_t$) source term contours of rotor

Figure 4.6.1: Base flow & Energy source term at 50% span, Circular HS at mid passage of IGV (not clocked), 21-21-21 blade count (Case 1)

Temperature contours of rotor base flow

Energy ($E_t$) source term contours of rotor

Figure 4.6.2: Base flow & Energy source term at 50% span, Circular HS at mid passage of IGV (not clocked), 21-28-21 blade count (Case 2)
Figure 4.6.3: Base flow & Energy source term at 50% span, Circular HS Impinging IGV (clocked), 21-21-21 blade count (Case 3)

Figure 4.6.4: Base flow & Energy source term at 50% span, Circular HS Impinging IGV (clocked), 21-28-21 blade count (Case 4)
Temperature contours of rotor base flow

Figure 4.6.5: Base flow & Energy source term at 50% span, Elliptical HS at mid passage of IGV (not clocked), 21-21-21 blade count (Case 5)

Temperature contours of rotor base flow

Energy ($E_t$) source term contours of rotor

Figure 4.6.6: Base flow & Energy source term at 50% span, Elliptical HS at mid passage of IGV (not clocked), 21-28-21 blade count (Case 6)
Temperature contours of rotor base flow

Energy ($E_i$) source term contours of rotor

Figure 4.6.7: Base flow & Energy source term at 50% span, Elliptical HS Impinging IGV (clocked), 21-21-21 blade count (Case 7)

Temperature contours of rotor base flow

Energy ($E_i$) source term contours of rotor

Figure 4.6.8: Base flow & Energy source term at 50% span, Elliptical HS Impinging IGV (clocked), 21-28-21 blade count (Case 8)
Figures 4.6 shows the temperature contours of rotor base flow and energy source term contours at 50% span for all the cases. In all the figures the energy source term contours have the same levels ranging from –2.0 to +2.0. It can be observed that the hot regions in figures 4.5, which show time average temperature contours, correspond to the positive energy source term contours in figures 4.6. Energy source term can be interpreted as a way to add or extract energy to the system. A positive energy source term will increase temperature in that region and a negative energy source term will decrease temperature.

4.1.1 Hot Streak Clocking Effects

The adjustment of circumferential locations of the first-stage stator blades with respect to the combustor hot streaks is known as hot streak clocking (Figure 4.1). Experiments and simulations have shown that hot streak clocking can be used to control rotor surface temperature.20,21,22 When hot streaks impinge on the leading edges of the IGVs the rotor surface temperatures have been shown to be significantly lower compared to cases in which the hot streaks are introduced mid-passage.

Simulations were performed for both the 1-1-1 & 3-4-3 blade count cases and for both circular & elliptical hot streak shapes. The hot streak clocking effect has been captured in all cases irrespective of blade count and hot streak shape.
Figure 4.7.1: IGV surface temperature profile, Circular HS, 21-21-21 blade count (Case1-Mid Passage & Case3-Impinging)
Figure 4.7.2: IGV surface temperature profile, Elliptical HS, 21-21-21 blade count
(Case10-Mid Passage & Case6-Impinging)
Figures 4.7 shows the comparison of IGV surface temperature profiles between mid passage & impinging hot streaks at various spanwise locations for both the circular hot streak (figure 4.7.1) and elliptical hot streak (figure 4.7.2). The effect of hot streak clocking in the IGV can be observed in figures 4.7. In the case of the hot streak impinging at the leading edge of the IGV, the IGV surface temperatures are very high in comparison to the mid passage hot streak case. The other phenomenon that can be seen is secondary flow. Since hot streaks in inviscid flow tend not to migrate as they convect through the IGV, the hub heating is primarily due to secondary flow. In Figure 4.8 it can be seen that hot streak is migrating towards the hub, these results are similar to the results by Dorney et al.}\(^{23}\)

![IGV surface temperature](image)

Figure 4.8: IGV surface temperature, Circular HS, Impinging IGV (Case3)
Figure 4.9.1: Time average rotor surface temperature profiles, Circular HS, 21-21-21 blade count (Case1-Mid Passage & Case3-Impinging)
Figure 4.9.2: Time average rotor surface temperature profile, Circular HS, 21-28-21 blade count (Case2-Mid Passage & Case4-Impinging)
Figure 4.9.3: Time average rotor surface temperature profile, Elliptical HS, 21-21-21 blade count (Case10-Mid Passage & Case7-Impinging)
Figures 4.9 shows the comparison of time average rotor surface temperature profiles between mid passage & impinging hot streaks at various spanwise locations for both blade counts and hot streak shapes. In each figure it can be seen that the rotor surface temperature in the impinging hot streak case is lower than in the mid passage hot streak case. This is due to the two reasons,

- In the case of the impinging hot streak, the hot streak is forced to mix with the cooler fluid of the stator wake. This acts to dissipate the hot streak and reduce its maximum temperature.

- Second, the hot streak now has a tendency to move towards the suction surface of the rotor along with the stator wake. The dissipated hot streak is rapidly convected along the suction side of the rotor passage before it can do any damage.

In figure 4.10 it can be observed that the suction surface temperature in the case of the impinging hot streak is slightly lower compare to mid passage hot streak. This is because the hot streak now mixes with the stator wake and convected towards the suction surface of the rotor.
Figure 4.10.1: Time average rotor pressure surface temperature, Circular HS, 21-21-21 blade count

Figure 4.10.2: Time average rotor suction surface temperature, Circular HS, 21-21-21 blade count
4.1.2 Hot Streak Shape Effects

Modifying the combustor fuel nozzle can change the shape of the hot streaks or temperature non-uniformities. Different hot streak shapes can have different blade surface heating. Two hot streak shapes have been studied, circular & elliptical. The hot fluid cross-sectional area of both shapes is same so that the average total enthalpy introduced by the hot streak is same in both cases. Figures 4.11 & 4.12 shows the comparison between the hot streak shapes for the mid passage hot streak and the 21-21-21 blade count. As the temperature ratio in case 5 & case 6 is 2.0 instead of 1.5 so case 9 & case 10 (fine grid) is considered for this study.

IGV surface temperature is plotted in figures 4.11.1 & 4.12.1, it can be seen that circular hot streak is gives more heating on the vane surface near mid span. Near the leading edge at 10% span & 25% span elliptical hot streak gives slightly higher heating, this might be due to the fact that the elliptical hot streak is longer in spanwise direction, it extends from 17% span to 63% span, whereas circular hot streak extends from 23% span to 57% span.

It can be observed from figures 4.11.2 & 4.12.2 (rotor surface temperature) that near the hub both hot streak shapes give the same amount of heating but away from the hub the circular hot streak leads to more blade surface heating, especially in the top half of the span. The reason for this effect is the circumferential temperature variation; in the circular hot streak case the circumferential variation of temperature is greater compare to the elliptical hot streak (See discussion on page 10) hence the preferential heating of the rotor pressure surface will be greater in the case of circular hot streak. IGV surface temperatures are also higher due to the same reason.
Figure 4.11.1: IGV pressure surface temperature, Impinging hot streak, 21-21-21 blade count

Circular Hot Streak (Case 3)  Elliptical Hot Streak (Case 7)

Figure 4.11.2: Time average rotor pressure surface temperature, mid passage, 21-21-21 blade count

Circular Hot Streak (Case 9)  Elliptical Hot Streak (Case 10)
Figure 4.12.1: IGV surface temperature profile, Impinging hot streak, 21-21-21 blade count (Case3-Circular HS & Case7-Elliptical HS)
Figure 4.12.2: Time average rotor surface temperature profile, Mid passage hot streak, 21-21-21 blade count (Case9-Circular HS & Case10-Elliptical HS)
4.1.3 Blade Count Ratio Effects

It is common practice to modify blade count in unsteady, 3-D turbomachinery simulations to avoid the necessity of solving all the passages in each row, but the effect of this assumption on hot streak results has not been evaluated. Though by using linearized method simulations for actual experimental configuration can be performed using only single passage in each row (as described earlier in section 2.1), but these different blade count ratio studies were performed because of the nonlinear simulations for same cases being performed by Dr. D. Sondak at Boston University as part of a larger project to study the effects of airfoil clocking on stage efficiency (nonlinear simulations only).

The experimental configuration has 22 airfoils in the first-stage stator row and 28 airfoils in each of the rotor and second-stage stator rows. To simulate this exact airfoil count, it would be required to model 11, 14 and 14 passages (for nonlinear simulations only) in the three blade rows respectively. For one set of simulations, the first blade row was modeled as having 21 airfoils, so the first-row geometry was scaled by 22/21 to maintain the correct blockage. In a similar fashion, the second and third blade row geometries were scaled by 28/21. This resulted in a blade count of 21-21-21 for the three blade rows, which can be modeled with one passage per row (1-1-1). For the second blade count, the first row geometry was scaled by 22/21 and the third row geometry was scaled by 28/21 to simulate a blade count of 21-28-21 (3-4-3). The IGV airfoil count in the 1-1-1 case was chosen equal to that in the 3-4-3 case so that the hot streaks would introduce the same amount of energy into the domain in both cases.

Figures 4.13 show the comparison of time average rotor temperature profiles at various
spanwise locations between two blade counts. The results for both hot streak shapes are presented circular hot streak (Case 1(21-21-21) & Case 2(21-28-21)), figure 4.12.1 and elliptical hot streak (Case 5(21-21-21) & Case 6(21-28-21)), figure 4.12.2. Please note that case 5 & case 6 are with a temperature ratio of 2.0 instead of 1.5. For a given number of hot streaks, it would be expected that more rotor blades will result in lower rotor blade temperatures, since the hot streaks are distributed among more rotor passages. And this effect is indeed observed in both figures 4.12.1 and 4.12.2. The differences are particularly large near the end walls, with relatively small differences at midspan.
Figure 4.13.1: Time average rotor surface temperature profile, Circular HS at mid passage (Case1-blade count 21-21-21 & Case2-blade count 21-28-21)
Figure 4.13.2: Time average rotor surface temperature profile, Elliptical HS at mid passage (Case5-blade count 21-21-21 & Case6-blade count 21-28-21)
4.1.4 Hot Streak Temperature Ratio Effects

Figure 4.14.1 shows the exit tangential flow angle of inlet guide vane for both hot streak temperature ratio. The differences between the two are relatively small except in the top half of span where difference is approximately 1.0 degree. These differences may be there due to the different grids used in both the cases (See the next section for detail).

Figure 4.14.1: Exit tangential flow angle, Inlet guide vane, Elliptical HS at mid passage

Figures 4.14.2 show the time averaged rotor surface temperature at various spanwise locations. As expected the higher temperature ratio case has more blade heating compare to lower temperature ratio. It is interesting to note that the difference is larger near the mid span compare to the end walls. The spikes on the HS temperature ratio 2.0 case (Case 5) may be due to the difference of the grids used in both the cases (See next section for details).
Figure 4.14.2: Time average rotor surface temperature profile, Elliptical HS at mid passage, 21-21-21 blade count (Case 10 HS temp. ratio=1.5 & Case 5 HS temp. ratio=2.0)
4.2 Grid Resolution

After comparison of the results from cases 1 to 4 with non-linear results\textsuperscript{27} it was observed that the results in hub region and near the rotor blade surface doesn’t match that well. The reason for this can be poor grids, with slightly bad $y^+$, so more cases were studied with better grid to study the grid resolution effect. Two cases with fine grids are presented over here, case 9 – circular HS at mid passage with 21-21-21 blade count & case 10 – elliptical HS at mid passage with 21-21-21 blade count.

The grid size for IGV was increased to 53 X 53 X 129 from 33 X 45 X 129, so as the $y^+$ at the hub are better and for rotor the grid size was kept the same but the $y^+$ at rotor surfaces were improved.

Figure 4.15 shows the mid span pressure profiles for IGV and rotor and they are compared with the experimental results of Dring et al. The IGV pressure profile is similar for both the grids but for the fine grid rotor pressure profile matches the experimental results more closely.

Figure 4.16 shows the exit tangential flow angle for IGV and rotor. The differences in exit flow angle for IGV are very small except around 60% to 80% span but for rotor the differences are very huge.
Figure 4.15: Mid span pressure profiles for IGV & Rotor

Figure 4.16: Exit tangential flow angle
Figure 4.17 shows the time average rotor surface temperature profiles for various spanwise locations. Near the end walls the profile is nearly same except at the trailing edge where some spikes in temperature profiles can be seen for poor grids. Also near mid span poor grid gives the higher heating compare to fine grid and on the suction surface near mid span it again gives some sort of temperature fluctuations. Hence it can be said that poor grid is insufficient to resolve the flow properly at the trailing edge of the rotor and fine grid is needed for that purpose.

Figure 4.17.1: Time averaged rotor surface temperature, circular HS at mid passage with 21-21-21 blade count
Figure 4.17.2: Time average rotor surface temperature profile, Circular HS at mid passage, 21-21-21 blade count (Case 1 poor grid & Case 10 fine grid)
5 Conclusions and Recommendations

5.1 Conclusions

- Hot streak fluid migrates toward the pressure surface of the rotor and colder fluid toward the passage.

- The pressure surface of the rotor exhibits highly elevated temperatures.

- Clocking hot streaks such that they impinge on the IGV leading edge results in generally lower rotor surface temperatures.

- Hot streak shapes with higher circumferential variation, for e.g. circular compared to elliptical shape, will have higher rotor surface temperatures.

- Scaling airfoil geometry to model different blade counts has a significant effect on predicted rotor surface temperatures. The difference is moderate at mid span and large at both end walls.

- Hot streak with higher temperature ratio have higher rotor surface temperature with larger temperature difference near the mid span compare to end walls.

- A grid with proper $y^+$ is needed to resolve the flow at the exit of rotor passage.

- Lumped deterministic source terms approach can be used to in modeling hot streaks.
with sufficient accuracy in considerably less amount of time.

## 5.2 Recommendations

- Hot streak clocking can be used to reduce the rotor surface temperatures.

- Hot streak shape with lower circumferential variation (e.g. elliptical compared to circular) is useful in reducing rotor surface temperatures.

- Hot streak shape and blade count effects can be used together to reduce the rotor surface temperature to a much lower values.

- It was noticed that the rotor surface temperatures are slightly lower than what are predicted by non-linear approach\(^\text{27}\). Upon due investigation it was found that temperatures at IGV surface matches pretty nicely but due to the coarse nature of grid after the trailing edge of IGV and second order spatial accuracy of the code the peak temperature is dissipated to a lower value at IGV exit. Also rotor inlet conditions are taken from IGV exit rather then interpolation of rotor inlet grid in IGV passage, so those lower temperatures due to additional dissipation are passed on to the rotor and hence resulting in lower rotor surface temperatures. To correct this problem either the grid after the trailing edge of IGV should be modified or the flow variables at rotor inlet plane should be interpolated from the IGV passage.
6 Future Work

- Rotor computations for one case should be performed by using interpolated values of
  flow variables from IGV passage rather than considering IGV exit plane values, and
  then this results should be compared with the current ones.

- Performing the same steps as done for rotor, computations for second-stage stator
  should be done using time average (base flow + DST) rotor flow.

- More cases should be computed to study the hot streak shape effects in detail and
  then determine the best shape for fuel nozzles, so as to minimize the rotor surface
  temperatures.

- Computations for actual (22-28-28) blade count should be performed to study the
  variations in rotor surface temperatures from other blade counts.
References


