Two-Dimensional Inlet Temperature Profile Attenuation in a Turbine Stage

DANIEL J. DORNEY and ROGER L. DAVIS
Theoretical & Computational Fluid Dynamics Group
United Technologies Research Center
East Hartford, CT 06108
and
OM P. SHARMA
Turbine Group
Pratt & Whitney
East Hartford, CT 06108

ABSTRACT

Experimental evidence has shown that hot streaks which are emitted from the combustors of gas turbines are often largely responsible for the burning of first stage turbine blades. Designers have attempted to counteract the effects of these hot streaks through the use of complex internal and film cooling schemes. Unfortunately, due to the lack of accurate predictive tools which account for temperature non-uniformities in the gas path, as well as a lack of detailed understanding of the physical mechanisms which control the migration and accumulation of the hot streak gases, turbine blade “hot spots” still occasionally occur. In an effort to increase understanding of the interaction mechanisms between combustor hot streaks and turbine blade heat transfer, a numerical investigation has been conducted to determine if a two-dimensional solution procedure can accurately predict rotor airfoil surface heating for flows which include planar hot streaks. A two-dimensional Navier-Stokes analysis is used to predict unsteady viscous flow through a 1-stator/1-rotor configuration with a planar hot streak introduced at the stator inlet. Comparison of the predicted results with a new experimental data set demonstrates that the two-dimensional numerical procedure can be used to accurately predict time-averaged rotor pressure surface temperatures for flows which include planar hot streaks.

Nomenclature

\begin{align*}
  a & \quad \text{Speed of sound} \\
  e & \quad \text{Specific energy} \\
  e_t & \quad \text{Total energy} \\
  J & \quad \text{Jacobian of transformation} \\
  M & \quad \text{Mach number} \\
  P & \quad \text{Static pressure} \\
  Pr & \quad \text{Prandtl number} \\
  R & \quad \text{Universal gas constant} \\
  Re & \quad \text{Free stream inlet reference Reynolds number} \\
  T & \quad \text{Static temperature} \\
  u, v & \quad x, y \text{ components of velocity} \\
  U & \quad \text{Rotor velocity} \\
  \beta & \quad \text{Rotor inlet relative flow angle} \\
  \kappa & \quad \text{Thermal conductivity} \\
  \mu, \lambda & \quad \text{First and second coefficients of viscosity} \\
  \rho & \quad \text{Density} \\
  r & \quad \text{Shear stress} \\
  c & \quad \text{Cold flow} \\
  HS & \quad \text{Hot streak} \\
  i & \quad \text{Inviscid} \\
  L & \quad \text{Laminar quantity} \\
  t & \quad \text{Stagnation quantity} \\
  T & \quad \text{Turbulent quantity} \\
  v & \quad \text{Viscous} \\
  w & \quad \text{Wall value} \\
  x, y & \quad \text{First derivative with respect to } x \text{ or } y \\
  xx, yy & \quad \text{Second derivative with respect to } x, y \\
  1 & \quad \text{Inlet quantity} \\
  2 & \quad \text{Exit quantity}
\end{align*}

INTRODUCTION

In recent years, design engineers have achieved substantial increases in turbine stage efficiencies by increasing combustor operating temperatures. The increased temperature of the combustor exit flow has necessitated the implementation of complex cooling schemes to avert burning of first stage turbine airfoils. In an effort to optimize the various cooling schemes, engineers have simulated (both experimentally (Butler et al, 1989) and compu-
tationally (Krouthen and Giles, 1988, Dorney et al, 1990, Takahashi and Ni, 1990, Sharma et al, 1990, Rai and Dring, 1990)) the “hot streaks” which exit the combustor and migrate through the turbine stage. The goal of these simulations has been to isolate the various factors associated with combustor hot streaks which lead to excessive temperatures and burning of first stage turbine blade pressures.

The work of Butler et al (1989) represents a detailed experimental investigation of hot streak migration. In this experiment, a three-dimensional hot streak was introduced into the stator inlet of a turbine stage through a circular pipe. The hot streak was seeded with CO₂, and the path of the hot streak was tracked by sampling the CO₂ concentrations at various locations in the turbine stage. According to Butler et al (1989), the experimental simulations indicate that hot streak migration paths and rotor surface heating are controlled by a) the secondary flows in the rotor passage, and b) the segregation of hot and cold gases caused by the difference in the rotor inlet flow angle between the hot and cold gases.

Attempts to reproduce the circular hot streak experimental results of Butler et al (1989) using two-dimensional computational techniques have only been partially successful (Krouthen and Giles, 1988, Dorney et al, 1990, Rai and Dring, 1996). The discrepancies between the predicted two-dimensional results and experimental data are probably due to a combination of factors, including a) the effects of secondary and endwall flows, b) the three-dimensional shape of this particular experimental hot streak, c) the assumed correlation between CO₂ (used in the experiments to track the movement of the hot streak) and temperature, d) the hot streak to rotor passage count, and e) the ratio of the hot streak temperature to that of the surrounding flow.

Some progress has been made in isolating the role of three-dimensional and viscous effects on hot streak migration (Dorney et al, 1990, Takahashi and Ni, 1990, Sharma et al, 1990) and turbine rotor heat transfer. Both a three-dimensional Navier-Stokes (Dorney et al, 1990) and an Euler simulation (Takahashi and Ni, 1990, Sharma et al, 1990) in which viscous effects were modelled have demonstrated better agreement with the circular hot streak experimental data (Butler et al, 1989) than previous two-dimensional simulations for the time-averaged temperature on the rotor pressure surface.

The goal of the current research effort is to determine if two-dimensional unsteady rotor/stator/hot streak simulations can provide reliable warnings as to when rotor surface temperatures become too high due to combustor hot streaks. If two-dimensional simulations can be shown to be useful in providing this guidance, then the need to execute costly and time-consuming three-dimensional simulations can be reduced or eliminated. Computational results for a two-dimensional hot streak simulation are compared with new experimental data for a two-dimensional planar hot streak which was introduced across the span of a turbine stage stator inlet. An important mechanism which has been identified as a controlling factor on the surface temperature of turbine rotors is hot streak shape. By understanding the strengths and limitations of two-dimensional hot streak simulations, useful insights can be obtained which will improve turbine blade designs.

Results of a 1-stator/1-rotor hot streak migration simulation of the Large Scale Rotating Rig (LSRR) (Butler et al, 1989) are compared with experimental data recently obtained for a planar hot streak. The close agreement between the numerical results and the experimental data suggests that two-dimensional numerical analyses can be used to accurately predict rotor pressure surface temperature distributions.

**NUMERICAL INTEGRATION PROCEDURE**

The governing equations of motion considered in this study are the time dependent two-dimensional Navier-Stokes equations, which can be written in Cartesian coordinates as:

\[ U_t + (F_x + F_y)_x + (G_x + G_y)_y = 0 \]  (1)

where

\[ U = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ e_t \end{bmatrix} \]

\[ F_x = \begin{bmatrix} \rho u \\ \rho u^2 + P \\ \rho uv \\ (e_t + P)u \end{bmatrix}, \quad F_y = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{xy} \end{bmatrix} \]

\[ G_x = \begin{bmatrix} \rho v \\ \rho vu \\ \rho v^2 + P \\ (e_t + P)v \end{bmatrix}, \quad G_y = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{yy} \end{bmatrix} \]

where

\[ \tau_{xx} = 2\mu\frac{\partial u}{\partial x} + \lambda\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right) \]

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

\[ \tau_{yy} = 2\mu\frac{\partial v}{\partial y} + \lambda\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right) \]

\[ \tau_{xx} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \gamma \mu P r^{-1} \epsilon_t \]

\[ \tau_{yy} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \gamma \mu P r^{-1} \epsilon_t \]

\[ \epsilon = \frac{P}{(\rho(\gamma - 1))} \]

\[ \epsilon_t = \rho e + \frac{\rho(u^2 + v^2)}{2} \]

For the present application, the second coefficient of viscosity is calculated using Stokes' hypothesis, \( \lambda = 2/3\mu \). The equations of motion are completed by the perfect gas law.

The viscous fluxes can be simplified by incorporating the thin layer assumption (Baldwin and Lomax, 1978). The thin layer
assumption states that for high Reynolds number flows, the diffusion terms normal to a solid surface will be much greater than those parallel to the surface. In the current study, viscous terms are retained in the direction normal to the blade surface.

To extend the equations of motion to turbulent flows, an eddy viscosity formulation is used. Thus, the effective viscosity and effective thermal conductivity can be defined as:

\[ \kappa = \mu_L + \mu_T \]
\[ C_T = \frac{\mu_L}{P_L} + \frac{\mu_T}{P_T} \]  

(5)

The turbulent viscosity, \( \mu_T \), is calculated using the two-layer Baldwin and Lomax (1978) algebraic turbulence model.

The numerical procedure for the two-dimensional analysis consists of a time marching, implicit, third-order spatially accurate, upwind, finite difference scheme. The inviscid fluxes are discretized according to the scheme developed by Osher (Chakravarthy and Osher, 1982). The viscous fluxes are calculated using standard central differences. An alternate direction, approximate-factorization technique is used to compute the time rate changes in the primary variables. In addition, an inner Newton iteration is used to increase stability and reduce linearization errors.

GRID GENERATION

The current two-dimensional Navier-Stokes analysis uses multiple zonal grids to discretize the rotor/stator flow field and facilitate relative motion of the rotor (Rai et al, 1988, 1989). A combination of O- and H-grid sections are generated in the blade-to-blade direction extending upstream of the stator leading edge to downstream of the rotor trailing edge. Algebraically generated H-grids are used in the regions upstream of the leading edge, downstream of the trailing edge and in the inter-blade region. The O-grids, which are body-fitted to the surfaces of the airfoils and generated using an elliptic equation solution procedure, are used to properly resolve the viscous flow in the blade passages and to easily apply the algebraic turbulence model. Computational grid lines within the O-grids are stretched in the blade-normal direction with a fine grid spacing at the wall. Figure 1 illustrates the grid topology used in the current simulations.

BOUNDARY CONDITIONS

The theory of characteristics is used to determine the boundary conditions at the stator inlet and rotor exit. For subsonic inlet flow, the total pressure, tangential velocity component and the downstream running Riemann invariant, \( R_1 = u + \frac{P}{\rho^2} \), are specified while the upstream running Riemann invariant, \( R_2 = u - \frac{P}{\rho^2} \), is extrapolated from the interior of the computational domain. Inlet flow boundary conditions within the hot streak region are updated using Riemann invariants where the velocity and speed of sound reflect the static and stagnation temperature increase in the hot streak. The static and total pressure in the hot streak are assumed to be equal to that of the undisturbed inlet flow, corresponding to the experimental conditions. For subsonic outflow the pressure is specified while the tangential component of velocity, entropy, and the downstream running Riemann invariant are extrapolated from the interior of the computational domain. Absolute no-slip boundary conditions are enforced on stator airfoil surfaces, while relative no-slip boundary conditions are imposed along the rotor airfoil surfaces. Periodicity is enforced along the outer boundaries of the H-grids in the circumferential (\( \varphi \)) direction. In the present, study the flow is assumed to be adiabatic.

Dirichlet conditions, in which the time rate change in the vector \( \mathbf{U} \) of Eq. (2) is set to zero, are imposed at the patched boundaries of the O- and H-grids in the stator and rotor regions. The flow variables of \( \mathbf{U} \) at the zonal boundaries are explicitly updated after each time step by interpolating values from the adjacent grid. Because of the explicit application of the zonal boundary conditions, large time steps necessitate the use of more than one Newton iteration. The zonal boundary conditions are non-conservative, but for subsonic flow this should not affect the accuracy of the final flow solution. Further information describing the implementation of the boundary conditions can be found in Rai et al (1985, 1988, 1989, 1990).

RESULTS

A numerical investigation of hot streak migration has been conducted in which the predicted results from a two-dimensional Navier-Stokes procedure have been compared with experimental data. The turbine airfoil geometry used in this study consisted of the first stage of the UTRC Large Scale Rotating Rig (LSRR) (Dring et al, 1982, Butler et al, 1989) which includes 22 stator airfoils and 28 rotor airfoils. An accurate simulation of this configuration would require at least 11 stator airfoils and 14 rotor airfoils. For the current 1-stator/1-rotor simulation, a rescaling strategy was used to reduce the number of airfoils to 1 stator and 1 rotor. It was assumed that there were 28 stator airfoils and 28 rotor airfoils and the stator was scaled down by the factor (22/28). To replicate the test conditions of the experimental
study, a 65% axial gap was used between the stator and rotor airfoils. The numerical simulation was performed at the experimental stator inlet Mach number of $M = .051$. In both the experiment and the numerical simulation, the stator vanes were rotated down 7 degrees with respect to the tangential direction relative to the design point operating conditions. The experimental rotor rotation speed was 710 rpm. The Reynolds number in both the experiment and numerical simulation was 100,000 per inch and a pressure ratio of $P_2/P_1 = .9640$ was determined from the inlet total pressure and the static pressure measured in the trailing-edge plane.

In the experimental study, 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 (i.e. one hot streak for the entire wheel). The temperature of the hot streak was twice that of the surrounding inlet flow, whereas the hot streak static and stagnation pressures were identical to the free stream. The hot streak was seeded with CO$_2$ and the path of the hot streak determination by measuring CO$_2$ concentrations at various locations within the turbine stage using the blade surface static pressure taps. In the numerical simulation, one hot streak is introduced to the inlet of every stator passage in the form of a hyperbolic tangent (step-like) temperature profile. A hot streak temperature of 1.2 times that of the surrounding inlet flow was chosen for this investigation. Shear layer instabilities, caused by the step-like temperature profile, developed in the numerical simulations when the hot streak temperature was increased beyond 1.2 times that of the surrounding flow. The predicted temperature distributions of the current study can still be related to the experimental data using the temperature coefficient scaling technique developed by Sharma et al. (1990). In the 1-stator/1-rotor simulation, the hot streak was introduced over one quarter of the stator pitch and centered at midgap.

For the 1-stator/1-rotor simulation, the stator grid system was constructed with 101 x 31 (streamwise x tangential) grid points in the O-grid and 85 x 51 grid points in the H-grid. The rotor grid system was constructed with 101 x 31 grid points in the O-grid and 78 x 51 grid points in the H-grid. A total of 14,575 grid points were used in this simulation. The stator airfoils had an average $y^+$ value of 0.78, while the rotor airfoils had an average $y^+$ value of 0.70. Figure 1 illustrates the grid topology used in the numerical simulations, where every other grid point in the O-grid has been omitted for clarity.

The numerical simulations performed during this investigation were computed on a four processor Alliant FX-80 minicomputer. Typical calculations required .00191 seconds per grid point per iteration computation time. In the current study, approximately six cycles at 2000 iterations per global cycle were needed to obtain a time-periodic solution.

In the experiment, the flow coefficient was held at $\phi = u_c/U = .35$. The subscript 'c' refers to the fact that the flow coefficient is based on the inlet velocity in the regions outside the hot streak (i.e. cold flow). Within the hot streak, a velocity increase proportional to $\sqrt{T_w/T_\infty}$ causes a significant increase in the average flow coefficient, $\phi = u_c/U$. Thus, in the numerical simulation, where a hot streak is included in every passage, the effect of this velocity increase within the hot streak on the flow coefficient must be taken into account. In addition, previous experimental data (Dring et al. 1982) and numerical simulations indicate that at a flow coefficient of $\phi = .35$ the flow separates on the pressure surface of the rotor. To avoid the additional complications that flow separation may present in interpreting the numerical results, the flow coefficient in the numerical simulation was increased until the time-averaged pressure side separation disappeared (approximately $\phi_c = .385$). The difference in the flow coefficients equates to the numerical simulation being performed at approximately 5 degrees more positive incidence at the rotor inlet than in the experiment. This increase in incidence corresponds to only a 10% shift in the spanwise location (towards the tip) at which the numerical simulation was performed, to a position located at 60% span. At this low flow coefficient, a 50 deg variation in the incidence occurs from the hub to the tip of the rotor airfoil. This can be shown using an analytic expression which relates the rotor inlet relative flow angle to the flow coefficient (assuming a given blade geometry):

$$\cot \beta = \cot \alpha - \phi^{-1}$$

where $\alpha$ is the absolute flow angle at the stator exit. As Figure 2 illustrates, the rotor inlet flow angle changes very rapidly with flow coefficient for this particular geometry.

In order to understand the incidence variation at the rotor inlet caused by the hot streak and stator wakes, a study was performed to determine their relevance. By taking the derivative of eq. (6), an equation is obtained which relates the change in the rotor inlet relative flow angle to the change in the flow coefficient:

$$\Delta \beta = \Delta \phi/c_{\alpha}c^2\beta$$

where $\beta$ is the mean rotor relative inlet flow angle, $\Delta \phi$ is the variation in the flow coefficient due to a hot streak or stator wake, and $\Delta \beta$ is the change in the rotor inlet relative flow angle.

![Figure 2: Rotor Inlet Relative Flow Angle vs Flow Coefficient](image-url)
Figure 3: Rotor inlet flow angle and static temperature distributions

Equation (7) is useful in estimating the change in the rotor inlet relative flow angle as the rotor passes through the hot streak. For the present numerical simulation, eq. (7) predicts a 4.12 deg variation in the rotor inlet relative flow angle due to the hot streak. Figure 3 shows the numerically predicted rotor inlet relative flow angle and static temperature for one cycle, where a cycle corresponds to the rotor blade rotating through an angle of $\frac{2\pi}{N}$ where $N$ is the number of stator blades (i.e. $N=28$ for the present simulations). The stator wake has little effect on the rotor inlet static temperature, but has a significant impact on the relative flow angle. The wake causes the rotor inlet relative flow angle to vary by approximately 4.9 deg in the direction of more negative incidence and affects the flow angle for nearly 50% of the cycle.

The hot streak is seen to cause a variation in both the static temperature and relative inlet flow angle. The hot streak causes the rotor inlet relative flow angle to vary by approximately 3.9 deg (compared to 4.12 deg calculated above) in the direction of more positive incidence and affects the flow angle for about 50% of the cycle. Thus, through the course of one cycle the rotor inlet relative flow angle varies by almost 10 deg, although the incidence variation in the rotor caused by the stator wake is nearly cancelled by that corresponding to the hot streak for time-averaged flow. Figure 3 also shows that the flow angle variation due to the hot streak lags the temperature increase due to the hot streak.

Figure 4 illustrates the predicted stator surface time-averaged pressure coefficient distribution. Unfortunately, no experimental time-averaged pressure data was available for the stator airfoils. The time-averaged pressure coefficient is defined as:

$$C_p = \frac{2(P_{avg} - P_i)}{\rho_i U^2}$$

where $P_{avg}$ is the local time-averaged pressure, $P_i$ is the inlet total pressure, $\rho_i$ is the average inlet free stream density, and $U$ is the rotor velocity. Figure 5 shows the predicted and experimental (Dring et al, 1982) rotor surface time-averaged pressure coefficient distributions. While the experimental data of Dring et al (1982) is for rotor/stator flow without a hot streak, past simulations (Dorney et al, 1990, Rai and Dring, 1990) have shown that for the present type of simulation, the pressure and temperature fields are decoupled. Excellent agreement exists between the predicted results and the experimental data for the rotor, except near the suction surface leading edge where the numerical simulation predicts a strong overspeed. This discrepancy is probably due to the slight difference (1 deg) in the time-averaged incidence to the rotor between the numerical simulation and the experiment (Dring et al, 1982).

The migration path of the numerical hot streak is displayed in Fig. 6, which shows static temperature contours at one instant in time during the course of a cycle. The hot streak is undisturbed as it migrates through the stator passage, except that the width of the hot streak decreases due to flow acceleration. The geometry of the rotor blade is such that the hot streak first impinges upon the pressure surface, then wraps around the leading edge and moves along the suction surface. The hot fluid on the suction surface is quickly convected downstream, while the hot fluid near the pressure surface remains in the rotor passage for some time after the rotor passes through the hot streak. The hot streak assumes a V shape (due to the difference in convection
speeds between the pressure and suction sides of the passage) as it continues to move downstream through the rotor passage, before breaking up and leaving the turbine in the form of discrete temperature eddies.

Figure 7 illustrates the experimental time-averaged CO$_2$ concentration contours for the rotor surface. The movement of the hot streak fluid from leading edge to the trailing edge of the pressure surface at approximately constant spanwise locations indicates the two-dimensional nature of the hot streak. Conspicuously absent is the radial migration of the hot streak on the pressure surface observed both experimentally (Butler et al, 1989) and in previous numerical simulations (Dorney et al, 1990, Takahashi and Ni, 1990) for a circular hot streak. Figure 8 compares the predicted time-averaged temperature coefficient distribution along the surface of the rotor to the experimental data taken at midspan. The temperature coefficient, $C_T$, is defined as (Takahashi and Ni, 1990, Sharma et al, 1990):

$$C_T = \frac{T - T_1}{T_{avg,le} - T_1}$$

where $T$ is the local time-averaged temperature and $T_{avg,le}$ is the time-averaged temperature at the rotor leading edge. Results of the numerical simulation show fair agreement with the planar hot streak experimental data. The numerical simulation underpredicts the temperature near the pressure surface leading edge, and also predicts a rapid temperature fall-off on the suction surface which is not observed in the experimental data. Figure 9 shows the time-averaged temperature contours for the entire rotor passage. The accumulation of hot streak fluid on the pressure side of the passage is evident. A close-up of the rotor pressure surface leading edge region (see Figure 10) shows that a region of cooler fluid is trapped between the hot streak fluid and the rotor surface. Similar results were reported in Dorney et al (1990), for a two-dimensional hot streak simulation at different operating conditions. Figure 11, which shows time-averaged temperature profiles at three locations along the pressure surface of the rotor, also indicates the presence of a cooler region near the surface at the leading edge. Time-averaged boundary layer edge...
Figure 10: Time-averaged temperature contours near the rotor pressure surface leading edge

locations are also included in Figure 11. At 25% axial chord the maximum temperature in the boundary layer occurs off the surface, approximately one-third the distance to the boundary layer edge. Similarly, at 50% axial chord the maximum temperature in the boundary layer occurs well above the rotor surface. At 75% axial chord, however, the time-averaged boundary layer is noticeably thinner and the maximum time-averaged temperature occurs at the rotor surface. A general observation drawn from this investigation is that as the boundary layer becomes thinner, the maximum temperature in a given boundary layer profile moves closer to the airfoil surface.

The differences between the experimental data and the predicted surface temperatures shown in Fig. 8 may be due to one of several factors. As shown in Dorney et al (1990), Takahashi and Ni (1990), and Sharma et al (1990) a significant effect on the surface temperature exists (for certain hot streak shapes) due to the three-dimensionality of the rotor surface boundary layers and the influence of secondary flows. A second factor which may also contribute to the discrepancies between the experimental data and the two-dimensional simulation is the assumed correlation between the predicted surface temperature and the experimentally measured CO$_2$ concentrations. In the experiment, the CO$_2$ concentrations along the surface of the rotor were determined by drawing samples of the gas in through static pressure taps. The suction force may cause CO$_2$ gas from well above the airfoil surface to be included in the sample. To test this hypothesis, the time-averaged temperature coefficient was redefined as:

\[
\hat{C}_T = \frac{T_M - T_i}{T_{avg,le} - T_i}
\]

where $T_M$ is the time-averaged temperature, area-averaged over a given boundary layer profile. Figure 12 compares the predicted results using the modified temperature coefficient definition with the experimental data. Excellent agreement now exists between the predicted and experimental results. A comparison of Figures 8 and 12 reveals that the predicted surface temperature is approximately equal to the area-averaged temperature in the boundary layer, except in the leading edge region. Figure 12 shows that two-dimensional simulations can be used to provide guidance as to when and where rotor surface temperatures exceed allowable limits, provided an averaged temperature across the boundary layer is examined. This is an important finding which shows that costly and time consuming three-dimensional simulations of hot streak migration may not need to be executed to determine if rotor pressure surface burning is likely. Further investigation is required to determine if this manner of examining average viscous layer temperatures is reliable and applicable to other flow conditions and geometries.
CONCLUSIONS

A numerical investigation has been performed to determine if two-dimensional stator/rotor/hot streak interaction simulations can provide reliable warnings to turbine designers as to when and where rotor surface temperatures exceed allowable limits. During this investigation, the results from an unsteady 1-stator/1-rotor/1-hot streak Navier-Stokes simulation were compared with new experimental data taken from a turbine stage which included a planar hot streak introduced at the stator inlet. The predicted and experimental rotor surface temperature distributions show close agreement, provided the proper correlation is made between the experimental CO2 concentrations and the numerically predicted temperatures. Thus, for certain hot streak shapes, two-dimensional numerical procedures can be used as accurate and efficient tools by designers attempting to optimize cooling systems.

Future work will focus on developing improved techniques for comparing numerically predicted temperature distributions with experimental data. Based on the agreement between the current numerical results with the available experimental data, another investigation will be performed which uses the two-dimensional Navier-Stokes procedure to identify the effects of rotor inlet relative flow angle and boundary layer thickness on hot streak migration and rotor pressure surface heating.

ACKNOWLEDGEMENTS

This work was supported by the United Technologies Research Center under the Corporate Research Program. The authors appreciate the helpful discussions with Robert Dring and Dave Joslyn of the United Technologies Research Center.

REFERENCES


