Computational Modeling and Thermal Paint Verification of Film-Cooling Designs
For an Unshrouded High-Pressure Turbine Blade

Steven G. Gegg *
Nathan J. Heidegger †
Ronald A. Mikkelsen ‡
Rolls-Royce Allison
P.O. Box 420
Indianapolis, Indiana 46206-0420

ABSTRACT

High pressure turbine blades are exposed to an extreme high temperature environment due to increasing turbine inlet temperature. High heat fluxes are likely on the blade pressure surface. Other regions, such as the trailing edge and blade tip may be difficult to cool uniformly. Unshrouded blades present an additional challenge due to the pressure driven transport of hot gas across the blade tip. The blade tip region is therefore prone to severe thermal stress, fatigue and oxidation. In order to develop effective cooling methods, designers require detailed flow and heat transfer information.

This paper reports on computational aerodynamics and heat transfer studies for an unshrouded high pressure turbine blade. The emphasis is placed on the application of appropriate 3-D models for the prediction of airfoil surface temperatures. Details of the film cooling model, boundary conditions and data exchange with heat transfer models are described. The analysis approach has been refined for design use to provide timely and accurate results. Film cooling designs are to be tailored for the best coverage of the blade tip region. Designs include near-tip pressure side films and blade tip cooling holes. Hole placement and angle are investigated to achieve the best coolant coverage on the blade tip. Analytical results are compared to a thermal paint test on engine hardware. In addition to film cooling strategies, other aerodynamic/heat transfer design approaches are discussed to address the cooling requirements for an unshrouded blade.

NOMENCLATURE

| Symbol | Description |
|--------|-------------|
| C      | Celsius degrees |
| C_t    | true chord |
| C_s    | axial chord |
| F      | Fahrenheit degrees |
| HPT    | high pressure turbine |
| Re     | Reynolds number |
| RIT    | rotor inlet temperature |
| T      | temperature |
| T_u    | turbulence intensity |
| TBC    | thermal barrier coating |
| U      | velocity magnitude |
| \( \vec{v} \) | coolant velocity vector |
| \( W_c \) | compressor core flow |
| \( c \) | local coordinate direction |
| \( h \) | convective heat transfer coefficient |
| \( \vec{n} \) | surface normal vector |
| \( r \) | radial coordinate |
| \( s \) | surface distance coordinate |
| \( u, v \) | velocity components |
| \( x \) | axial coordinate |

Greek Symbols

| Symbol | Description |
|--------|-------------|
| \( \alpha \) | coolant flow angle with respect to surface normal |
| \( \rho \) | density |

*Development Engineer, Turbine Aerodynamic Design
†Project Engineer, Aerothermal Methods
‡Development Engineer, Heat Transfer

Presented at the International Gas Turbine & Aeroengine Congress & Exhibition
Indianapolis, Indiana — June 7—June 10, 1999
The design, development and manufacture of gas turbine engines is a complex process that requires from three to five years to advance from initial concept to full scale production. A significant part of this time is spent in the design and development of the hot section components. This time is needed to finalize the design details, procure tooling and manufacture the parts. Any design changes that become necessary in testing or through improved analyses or manufacturing techniques will delay the product introduction. The design engineer must carefully consider the technical information available from analytical solutions, experimental data and experience.

An overall view of the design process is illustrated in Figure 1. Clearly, design engineers require detailed information from multiple disciplines in order to develop hot section components. Expertise is required in aerodynamics, heat transfer, stress, dynamics, materials science and mechanical design.

The analysis approach described in this paper uses 3-D Computational Fluid Dynamics (CFD) modeling to predict the gas path temperatures for subsequent heat transfer analysis. Three-dimensional modeling of the gas path flow field captures important secondary flows which transport hot gas radially in the blade passage. The objective of introducing 3-D CFD into the design process is to increase the accuracy of flow and heat transfer predictions while at the same time reducing the time and cost of downstream activities.

The development that follows provides the motivation for introducing 3-D CFD modeling into the design phase. We describe the gas path flow to be modeled and explain the simplifications that provide 3-D results suitable for routine design use. Gas path temperature predictions from the 3-D CFD code are provided as boundary conditions for heat transfer calculations. The resulting predictions for blade surface temperature produce good agreement with thermal paint test results. The realistic predictions give design engineers confidence in current 3-D analytical tools and can ultimately offset some routine thermal paint tests, thus shortening both design and development time.

**OVERVIEW OF AERODYNAMIC FLOW FEATURES**

Figure 2 shows the flowpath and blading configuration for a modern air cooled high pressure turbine (HPT). We will focus attention on the unshrouded first stage blade. The unshrouded blade is a multi-pass convection cooled airfoil. Field experience for such a blade typically shows thermal distress at the tip. Tip distress is frequently attributed to tip rub. However, unshrouded blades that have not experienced a rub also provide evidence of high heat load at the tip. To help understand the possible causes for high heat load at the blade tip, 3-D CFD models have been used to simulate the gas path flow.

Solutions are obtained using a 3-D multi-row steady flow CFD simulation. Details of the 3-D CFD code are described in the next section. Some observations of the flow phenomena key to this study are described here. These flow features are also described in the earlier work of Roback and Dring (1992a, b); Saxer and Giles (1993); Gundy-Burlet and Dorney (1998).
CFD studies indicate that the thermal environment at the blade tip is influenced by upstream conditions: burner profile (radial and tangential), total pressure profile, coolant injection, and wakes. The present model was tuned to test data using a non-uniform HPT inlet total pressure profile. This derived pressure profile was necessary to simultaneously match thermocouple visualization studies confirmed that pressure side films would provide coolant coverage of the blade tip, provided sufficient coolant pressure margin could be obtained. CFD results as described above had been used successfully to improve heat transfer predictions in comparison to paint test results for non-film cooled airfoils. The work reported here extends such analyses to film cooling applications within design cycle time constraints.

CFD MODELING OF HOT GAS PATH

The modeling approach described below required an engineering analysis tool capable of coolant flow mass addition. An ad hoc approach to film coolant modeling was adopted as a first order design/analysis tool. Coolant flow rates, hole placement and coolant flow angle with respect to the blade surface are among the design choices that are considered.

ADPAC Navier-Stokes Numerical Algorithm

The aerodynamic predictions for the cases described in this study were obtained using ADPAC (ADVanced Propulsion Analysis Code). The ADPAC code is a general purpose turbomachinery aerodynamic design analysis tool which has undergone extensive development, testing, and verification (Hall et al., 1993). Detailed code documentation is also available for the ADPAC program (Hall et al., 1998). A brief description of the theoretical basis for the ADPAC analysis follows, and the interested reader is referred to the cited references for additional details.

The ADPAC analysis solves a time-dependent form of the three-dimensional Reynolds-averaged Navier-Stokes equations using a proven time-marching numerical formulation. The code employs proven numerics based on a finite volume, explicit multi-grid Runge-Kutta time-marching solution algorithm derived from the developmental efforts of Jameson et al. (1981); Adamczyk et al. (1989); Jorgensen and Chima (1989); and Ameri and Arnone (1992). Steady state flows are obtained as the time-independent limit of the time-marching procedure. Several steady-state convergence acceleration techniques (local time stepping, implicit residual smoothing, and multi-grid) are available to improve the overall computational efficiency of the analysis. The ADPAC code permits the use of a multiple-blocked mesh discretization which provides extreme flexibility for analyzing complex geometries. ADPAC offers the choice of three different turbulence models: the algebraic Baldwin-Lomax turbulence model (Baldwin and Lomax, 1978), the one-equation Spalart-Allmaras turbulence model (Spalart and Allmaras, 1992), and a two-equation k-ε turbulence model (Goldberg, 1994).

Mesh Generation

A computational mesh comprised of five mesh blocks was generated defining the turbomachinery geometry of the vane and blade. The five mesh blocks were a combination of O-mesh and H-mesh topologies as shown in Figure 4. Both the vane and the blade were meshed using an O-type mesh with 233 points around the blade, 65 points hub to case, and 21 points out from the blade surface to the periodic boundary. Two H-type meshes were used to extend the O-meshes upstream; the inlet mesh measured 5 x 65 x 21 and the inter-blade mesh 33 x 65 x 21. This inter-blade mesh was used in another design study to examine the influence of coolant injection immediately upstream of the blade tip. The final remaining mesh, a degenerate O-mesh, was used to model the tip clearance above the blade as shown in Figure 5. This mesh contains 233 points around the airfoil, 9 points radially in the clearance, and 5 points from the camber line to the tip edge.
Due to the shifted alignment of the mesh points between the tip clearance mesh and the blade body 0-mesh, a 1-D interpolation was performed across the contiguous boundary. Prior studies on company turbines with ADPAC have shown that the above mesh sizes yield grid independent solutions for aerodynamic quantities.

The unsteady interaction between the vane row and blade row was approximated by employing a “mixing plane” at the interface as shown in Figure 6. The mixing plane allows for the influence of the neighboring blade rows to be conveyed in an averaged sense. The values of the flow variables (i.e., \( p, p_u, p_v \), etc.) are averaged in the circumferential direction at the interface, and these averaged values are passed to the other side. This approximation allows the simulation to be run in a “steady-state” mode, rather than trying to resolve the time-periodic characteristics of the vane-blade interaction problem which would require significantly more computational time.

### Cooling Hole Injection Modeling

To accurately model film-cooling flow injected along the pressure side of a turbine blade, the computational mesh should be generated accounting for the exact placement and angle of the cooling holes. To perform this detailed grid generation may take weeks to months to represent all the fine details of this complex geometry intersection problem. An overview of previous work is presented by Kercher (1998). Recent literature contains several examples of discrete cooling hole injection modeling including both detailed hole analyses (Walters and Leylek, 1997; McGovern and Leylek, 1997; Hyams and Leylek, 1997; Brittingham and Leylek, 1997; Raman et al., 1998; Lin et al., 1997; Martin and Thole, 1997) and the incorporation into turbine airfoil geometries (Hall et al., 1994; Garg, 1997; Garg and Gaugler, 1994, 1997).

To be useful in a design cycle as described in Figure 1 with several redefinitions possible, approximations are needed to create a faster method of cooling hole location and modeling. The computational meshes used in this study were generated without prior knowledge of the exact cooling hole locations which allowed for faster mesh generation within the design cycle. Future development efforts could be directed at streamlining the direct incorporation of the cooling hole locations into the mesh generation process.

The cooling holes are modeled strictly through boundary conditions applied directly at the surface of the blade. The computational cells nearest the cooling hole outlet location were defined as inflow boundaries rather than viscous solid surfaces.
The number of cells used to represent the cooling holes was either one or two depending upon the cross-sectional area being matched. It is realized that in order to perform a detailed analysis of film-cooling several more cells are required to accurately model the injection profile of the cooling holes; however, the reader is reminded that compromises were implemented in order to facilitate the use of improved CFD predictions within the design cycle limitations. The design intent of the addition of film-cooling holes was to assist in the cooling of the blade tip. It is assumed that the reduction in computational detail of the individual holes, while important in the local characterization of the cooling effects, does not have a strong effect upon the convective travel of the cooling flow as it is entrained within the main passage flow up and over the blade tip. The total pressure specified at the boundary conditions for each of the six pressure-side film-cooling holes and the two tip cooling holes were iterated upon until the specified design cooling mass flow was achieved. Because the cooling holes were modeled using the closest cell approximation, the cross-sectional area of the cooling hole varied from the specified design area based on the cooling hole diameter. The variance in this area specification combined with the mass flow constraint produced a variation in exit flow velocities that may not represent actual velocities within the physical geometry. This was the result of a compromise to match cooling flow injection mass flow and use a quickly generated mesh at the expense of variations in injection velocity. In all cases the as-designed and as-modeled velocity ratios \( \left( \frac{U_c}{U_{\infty}} \right) \) for film cooling were in or very near the flow regime characterized by L'Ecuyer as the penetration regime (L'Ecuyer and Soechting, 1985). The effectiveness distribution in this case is dominated by excessive coolant penetration and augmented diffusivity due to jet/free stream interaction.

The direction of injection is controlled by two angle specifications in the boundary condition statement. These angles, represented in Figure 7 as \( \alpha_1 \) and \( \alpha_2 \), define the angle from the cell normal and the injection flow in the direction of the two local coordinate directions. That is, a value of zero for both of the angle descriptions would simulate injection flow normal to the
FIGURE 7: Flow injection angle defined by two angles between the normal vector and the respective computational index directions.

local blade surface. For the cases within this paper simulated with angled injection flow, $\alpha_1$ was set to 60 degrees and $\alpha_2$ was set to 45 degrees.

**Relevant Hardware and Simulation Parameters**

Quantities which help to characterize the hardware and simulation are given in Table 1. All solutions use the same CFD mesh along with the experimentally determined vane inlet temperature profile. The vane inlet pressure profile is also fixed according to initial studies to tune the CFD solutions to measured radial profiles of temperature downstream of the turbine stage. All solutions are computed at the same type test cycle condition unless otherwise noted.

Design values for the pressure side film coolant are shown in Table 2. The cooling holes are numbered from 1 - 6 with hole number 1 nearest the leading edge.

**Turbulence Model**

In the solution procedure developed here, the heat transfer coefficients ($h$) are calculated using a boundary layer analysis code. The subsequent finite-element heat transfer analysis only requires the near-surface gas temperatures computed for an adiabatic wall boundary condition by the 3-D flow solver. This lessens the requirements for extremely small near-wall spacings and higher-order turbulence models that would be necessary if the heat transfer coefficients were calculated directly from the predicted thermal gradients for a non-adiabatic condition. With this taken into consideration, the selection of the turbulence model should be based on a trade-off of accuracy versus computational expense. As the main emphasis of the current study is to incorporate these analyses into an iterative design cycle, the goal of reducing computational cost is important. Previous work investigating the impact of turbulence models upon the prediction of turbine blade heat transfer (Ameri and Arnone, 1994; Garg and Gaugler, 1997) with and without film-cooling indicated that the use of higher-order two-equation models (i.e., $k-\varepsilon, q-\omega$) "does not provide an overall more accurate solution" or only a "somewhat better" result than the zeroth-order Baldwin-Lomax model. The advantage of the Baldwin-Lomax model is magnified when the computational cost associated with the higher-order models (40% to 65% more cpu time) is taken into account. Therefore, the calculations presented within this paper employed the algebraic Baldwin-Lomax turbulence model with wall functions.

**Numerical Prediction Results**

A total of five different configurations were simulated using ADPAC: a baseline "no injection" case and four different cooling injection schemes. The cooling schemes allowed for two different sets of cooling sites: six pressure-side film cooling holes and/or two tip cooling holes on the tip cap near the trailing edge. From these available configurations, the turbine stage was modeled using the following four cooling schemes: normal pressure-side injection, angled pressure-side injection, tip hole cooling only, and angled pressure-side injection with tip hole cooling. The
detailed results presented within this section are referenced to the baseline "no injection" case; this should show where and to what degree the different cooling schemes are showing the largest impact. It is important to note that the thermal boundary condition modeled at the solid surfaces was an adiabatic wall; also, the comparisons and evaluations of the different cooling schemes within this section are strictly related to the impact upon the surface gas temperature and do not consider any benefit due to internal cooling passages.

A three-dimensional view of the blade tip, shown in Figure 8, shows a comparison between the no injection and the pressure-side injection (both normal and angled) without the tip holes injection. The blade surface has been shaded according to the surface gas temperature and particle traces have been released from the cooling sites to provide a qualitative evaluation. The particle traces show the injected cooling flow travel over the blade tip, through the tip clearance region, and become enveloped within the tip vortex. From the surface contours and the particle traces, it can be seen that the normal injection flow, because it has no stream-wise angle component, travels over the blade tip closer to the leading edge as compared to the angled injection flow. In the tip trailing edge region, the angled injection flow, due to its radial angled component was predicted not to cover the trailing edge as well as the normal injected cooling flow.

The surface gas temperature distribution can be best shown by "unwrapping" the 3-D blade geometry into a 2-D surface with coordinates of surface distance and radius (a-r). Whereas Figure 8 only presented the cooling effects on the pressure side of the blade, the unwrapped distribution of surface gas temperature shows the impact on all of the blade surface. Figure 9 shows the impact that each of the four cooling schemes has upon the baseline case. The turbine blade was unwrapped in this figure with the leading edge at the center of the figure, the pressure side to the left and the suction side to the right; the trailing edge of the blade is at the farthest left and farthest right extents. It is interesting to note that for the cases with the six pressure-side cooling injection sites, after the injected cooling flow travels over the blade tip it affects the suction-side gas temperature distribution for up to 40% of the upper blade span. The case where only the two tip holes were employed showed no significant cooling advantage upon either the pressure side or suction side of the blade.

To provide a more quantitative comparison, Figure 10 shows the predicted surface gas temperature distributions as extracted from the contours shown in Figure 9 along four different span-wise locations. Again the data are presented in an unwrapped setting such that the blade leading edge is centered at s = 0 and the pressure side is to the left (s < 0) and the suction side is to the right (s > 0). The discrete cooling sites can be clearly seen in the third plot from the top (94.0% blade span) as the sharp valleys in the temperature distribution; the two cooling sites closest to the leading edge are so close together that in this simulation the two cooling streams quickly merged into one moving radially outward. The normal injection cooling scheme (solid line) shows the greatest decrease in surface gas temperature at the leading edge and trailing edge of the pressure side.

Also, the normal injection scheme appears to provide a slightly better cooling coverage on the suction side of the blade surface. The configuration with only the two tip holes (dotted line) shows no change in surface gas temperature with the exception of the suction side trailing edge region at the near-tip location (99.6% blade span). The two configurations with angled cooling (dashed line and dash-dot line) are similar except in the region affected by the tip holes. At the lowest radial location presented (86.0% blade span) below the injection sites, there is a slight increase in predicted surface gas temperature possibly due to the increased blockage of the injected flow; however, again it needs to be noted that these predictions do not account for any conduction heat transfer from internal cooling of the blade.

The impact of the two tip cooling holes can be seen dramatically in the contours of change in surface gas temperature at the blade tip shown in Figure 11. In the two cases employing tip holes, the injected flow from the tip holes is shown to travel directly to the suction side of the blade and provide little surface cooling to the tip trailing edge. The distributions of change in
FIGURE 9: Shaded contours of changes in surface gas temperature from the baseline “no injection” case due to each of the cooling schemes.

FIGURE 10: Changes in surface gas temperature from the baseline “no injection” case extracted at four span-wise locations.
FIGURE 11: Shaded contours of changes in surface gas temperature from the baseline “no injection” case across the blade tip.

FIGURE 12: Changes in surface gas temperature from the baseline “no injection” case extracted along the tip camber line.

FIGURE 13: Blade geometry.
temperature and the coupon is immersed for a predetermined time. A full set of coupons is produced for the thermal paint operating range in 10°C (18°F) increments. The multi-change paints provide color transitions in the temperature range 240°C-1180°C (464°F-2156°F).

For each type of paint a specific label is assigned to each isotherm. A skilled technician compares the indicated thermal paint color transitions on the test article with calibration data. Correlations conducted between thermocouple and thermally painted components indicate that the maximum error between the two techniques is of the order ±20°C (±36°F) (Skelton, 1996).

The predicted temperatures for the blade were numerically solved using the 3-D finite element model shown in Figure 16. The numerical solution requires the definition of the boundary conditions (heat transfer coefficient and temperature) for each of the element faces exposed to either the gas (external) or the cooling air (internal) environment. A computer model of the cooling air passages inside of the blade was used to predict the air flow rate through the blade during engine operation. Each blade was flow tested during manufacture to check conformity with design specifications. The calculated air flow rate through the blade was used in experimentally determined correlations to determine the blade to cooling air heat transfer coefficients for the passages containing turbulators or fins. The cooling air temperature inside of the blade was calculated simultaneously with the solution of the blade metal temperatures.

The blade external heat transfer coefficients ($h_g$) and gas temperatures ($T_g$) were initially calculated using a 2-D computer code developed at Rolls-Royce Allison. The code requires as input a radial gas temperature profile out of the combustor (measured experimentally during testing of the combustor liner) and aerothermal data that defines the gas conditions for streamlines along the surface of the blade (determined analytically). The code computes $h_g$ and $T_g$ around the perimeter of the blade for eleven span-wise locations between the tip and the hub. The gas to metal heat transfer coefficients for the tip cap (top of the blade) were calculated using a flat plate turbulent flow correlation. The tip cap was divided into 18 segments (1 at the leading edge, 18 at the trailing edge). The gas velocity over each segment was calculated using the isentropic flow equations. The surface static pressure on the pressure side of the blade was the inlet pressure for the segment. The exit pressure for the segment was based on the surface static pressure on the suction side of the blade. The 2-D computer code provided the surface static pressures around the sides of the blade at the tip. The computed values of $h_g$ and $T_g$ are mapped onto the 3-D finite element model, thus defining the gas-to-blade heat flux.

The accuracy of the correlations used in predicting the gas-to-blade and blade-to-cooling air heat transfer coefficients is about ±15%. This translates into an accuracy of ±44°C (±79°F) in terms of predicted blade temperature (over-predicted $h_g$ with under-predicted $h_c$ and under-predicted $h_g$ with over-predicted $h_c$) assuming gas and cooling air temperatures are accurate.

The agreement between the calculated and measured temperatures was good over most of the airfoil. However, in evaluating the tip of a non-film cooled blade, it was found that temperature predictions could be brought into better agreement with measured values, if the gas boundary temperatures were defined using the 3-D CFD methods rather than the current 2-D code. A comparison of the 2-D and 3-D gas temperature results for the 80% span location on the blade is shown in Figure 14. The solution shown is for the non-film cooled blade. The 3-D CFD result not only captures the radial migration of hot gas but also indicates a segregation of the hotter and colder gas to the pressure and suction side of the passage respectively.

Another advantage of the 3-D CFD analysis used here is the ability to model the first order effects of coolant injection. Using the gas path CFD temperature predictions for cases with and without injection, the corresponding heat transfer predictions for surface temperature were computed. The predicted surface temperature benefit at the blade tip for the angled pressure side injection scheme is shown in Figure 15. This result indicates surface temperature reductions of $\approx 167°C$ (300°F) at mid chord on the pressure side. The predicted average surface temperature reduction at the blade tip is $\approx 72.2°C$ (130°F). An ensemble average of paint test data scaled to the same cycle point as shown in this figure indicates $\approx 50°C$ (90°F) overall surface temperature reduction due to pressure side coolant injection. Gas temperature boundary conditions for this result correspond to the solution for angled injection shown in Figure 16.

A solution of the thermal model using the 3-D CFD results for angled pressure side coolant injection is shown in Figure 16. This result is at a paint test condition corresponding to the engine hardware shown in Figure 17. The thermal paint results shown here were obtained for a blade with angled pressure side injection following a three minute paint test using a Rolls Royce thermal paint. Marked contours in Figure 17 correspond to the labeled contour scale used in Figure 16. The prediction lines up with under-predicted $h_c$ and under-predicted $h_g$ with over-predicted $h_c$ assuming gas and cooling air temperatures are accurate.
The benefits of pressure side film cooling are over predicted by the present heat transfer model. This solution represents the first effort to predict tip cap temperatures with the simple film coolant modeling provided by the 3-D CFD solution. Assuming that the blade internal coolant flow rate and temperature has been accurately modeled, this prediction suggests that it will be necessary to reevaluate the internal and external heat transfer coefficients in order to improve future predictions. However, as stated in the earlier discussion of Figure 15, the predicted reduction in tip cap temperature due to pressure side film cooling has been found to be representative of thermal paint results. Other key features present in the analytical result and the engine paint test are the "hot" spot on the pressure and suction sides of the blade at mid chord and approximately 60% span. This area of the blade is just downstream of the second partition (Figure 13). The hotter surface temperatures persist from this area back to the trailing edge at approximately 60% span.

CONCLUSIONS

The analytical approach and results presented in this paper provide reasonable agreement with thermal paint data. This approach is made fast and versatile through the use of a common 3-D mesh for the external flowfield. Coolant flow is added at prescribed flow angles via boundary conditions. The simple mass addition model described is shown to be relevant for near-tip pressure side film cooling schemes with coolant-to-free-stream velocity ratios in the penetration regime. A multi-row 3-D CFD solution using the simplified film cooling model then provides the gas path temperatures for a 3-D finite element heat transfer solution. Convection coefficients are obtained using the traditional 2-D design system methods with experience factors that match in-house data. This design approach provides technical insight into the first order effects of film coolant injection.

There are some practical modeling improvements that could be made to this approach which would not significantly impact the design cycle. One shortcoming of using an existing 3-D structured mesh to model the film coolant is that the coolant hole areas are not modeled exactly. Though the coolant flow rate, flow angle and temperature are correctly modeled, the coolant momentum is not exactly matched. A simple strategy to model the correct coolant momentum is to post-process the baseline structured mesh such that the existing mesh points are rearranged to simulate the correct coolant flow area. A practical, streamlined design/analysis approach as described here, combined with representative thermal paint testing, can provide acceptable engineering solutions.

ACKNOWLEDGMENTS

The authors wish to express their appreciation to Rolls-Royce Allison for granting permission to present this work. In addition, the authors gratefully acknowledge the technical support provided by Terry Moore, Philip Burkholder, Ed Hall, and Kurt Weher.

REFERENCES

Adamczyk, J. J., Celestina, M. L., Beach, T. A., and Barnett, M., 1989, “Simulation of Three-Dimensional Viscous Flow Within a Multistage Turbine,” ASME Paper 89-GT-152.

Ameri, A. A. and Arnone, A., 1992, “Navier-Stokes Heat Transfer Predictions Using Two-Equation Turbulence Closures,” AIAA Paper 92-3067.

Ameri, A. A. and Arnone, A., 1994, “Prediction of Turbine Blade Passage Heat Transfer Using a Zero and a Two-Equation Turbulence Model,” ASME Paper 94-GT-122.

Baldwin, B. S. and Lomax, H., 1978, “Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows,” AIAA Paper 78-257.

Brittingham, R. A. and Leylek, J. H., 1997, “A Detailed Analysis of Film-Cooling Physics, Part IV: Part IV: Compound-Angle Injection with Angled Holes,” ASME Paper 97-GT-272.

Ferguson, J. D., Walters, D. K., and Leylek, J. H., 1998, “Performance of Turbulence Models and Near-Wall Treatments in Discrete Jet Film Cooling Simulations,” ASME Paper 98-GT-438.

Garg, V. K., 1997, “Comparison of Predicted and Experimental Heat Transfer on a Film-Cooled Rotating Blade Using a Two-Equation Turbulence Model,” ASME Paper 97-GT-220.

Garg, V. K. and Gaugler, R. E., 1994, “Prediction of Film Cooling on Gas Turbine Airfoils,” ASME Paper 94-GT-16.

Gard, V. K. and Gaugler, R. E., 1997, “Effect of Velocity and Temperature Distribution at the Exit on Film Cooling of Turbine Blades,” Journal of Turbomachinery, Vol. 119, pp. 343-351.

Goldberg, U. C., 1994, “Towards a Pointwise Turbulence Model for Wall-Bounded and Free Shear Flows Boundary Layer and Free Shear Flows,” Asme fed-vol. 184.

Gundy-Burlet, K. L. and Dorney, D. J., 1998, “Effects of Radial Location on the Migration of Hot Streaks in a 1-1/2 Stage Turbine,” AIAA Paper 98-3578.

Hall, E. J., Heiddegger, N. J., and Delaney, R. A., 1998, “Task 15 - ADPAC User’s Manual,” NASA CR 206600.

Hall, E. J., Topp, D. A., and Delaney, R. A., 1994, “Aero-
FIGURE 16: Finite element model with predicted temperatures for thermal paint test.

FIGURE 17: Thermal paint results for the unshrouded HPT blade. Contour labels correspond to the legend in Figure 16.
dynamic/Heat Transfer Analysis of Discrete Site Film-Cooled Turbine Airfoils," AIAA Paper 94-3070.
Hall, E. J., Topp, D. A., Heidegger, N. J., and Delaney, R. A., 1995, "Energy Efficient Engine: High Pressure Compressor Detail Design Report," NASA CR 198465.
Hyams, D. G. and Leylek, J. H., 1997, "A Detailed Analysis of Film-Cooling Physics, Part III: Streamwise Injection with Shaped Holes," ASME Paper 97-GT-271.
Jameson, A., Schmidt, W., and Turkel, E., 1981, "Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes," AIAA Paper 81-1259.
Jorgensen, P. C. E. and Chima, R. V., 1989, "An Unconditionally Stable Runge-Kutta Method for Unsteady Flows," NASA TM 101347.
Kercher, D. M., 1998, "A Film-Cooling CFD Bibliography: 1971-1995," Int. Journal of Rot. Mach., Vol. 4, No. 1, pp. 61-72.
L'Ecuyer, M. R. and Soechting, F. O., 1985, "A Model for Correlating Flat Plate Film Cooling Effectiveness for Rows of Round Holes," in: Heat Transfer and Cooling in Gas Turbines: AGARD Conference Proceedings, no. 390, Bergen, Norway.
Lin, Y.-L., Stephens, M. A., and Shih, T. I.-P., 1997, "Computation of Leading-Edge Film Cooling with Injection through Rows of Compound-Angle Holes," ASME Paper 97-GT-298.
Martin, C. A. and Thole, K. A., 1997, "A CFD Benchmark Study: Leading Edge Film-Cooling with Compound Angle Injection," ASME Paper 97-GT-297.
McGovern, K. T. and Leylek, J. H., 1997, "A Detailed Analysis of Film-Cooling Physics, Part II: Compound-Angle Injection with Cylindrical Holes," ASME Paper 97-GT-270.
Roback, R. J. and Dring, R. P., 1992a, "Hot Streaks and Phantom Cooling in a Turbine Rotor Passage Part 1 - Separate Effects," ASME Paper 92-GT-75.
Roback, R. J. and Dring, R. P., 1992b, "Hot Streaks and Phantom Cooling in a Turbine Rotor Passage Part 2 - Combined Effects and Analytical Modelling," ASME Paper 92-GT-76.
Saxer, A. P. and Giles, M. B., 1993, "Predictions of 3-D Steady and Unsteady Inviscid Transonic Stator/Rotor Interaction with Inlet Radial Temperature Non-Uniformity," ASME Paper 93-GT-10.
Skelton, M., 1996, "Temperature Indicating Paint: Level 1 & 2 Core Technology Theory Package," Rolls-Royce PLC Internal Report.
Spalart, P. R. and Allmaras, S. R., 1992, "A One-Equation Turbulence Model for Aerodynamic Flows," AIAA Paper 92-0439.
Walters, D. K. and Leylek, J. H., 1997, "A Detailed Analysis of Film-Cooling Physics, Part I: Streamwise Injection with Cylindrical Holes," ASME Paper 97-GT-269.