HOST Turbine Heat Transfer
Program Summary

Herbert J. Gladden and Robert J. Simoneau
Lewis Research Center
Cleveland, Ohio

Prepared for the
33rd International Gas Turbine and Aeroengine Congress and Exposition
sponsored by the American Society of Mechanical Engineers
Amsterdam, The Netherlands, June 5–9, 1988
HOST TURBINE HEAT TRANSFER PROGRAM SUMMARY

Herbert J. Gladden and Robert J. Simoneau
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

ABSTRACT

The objectives of the HOST Turbine Heat Transfer subproject were to obtain a better understanding of the physics of the aerothermodynamic phenomena and to assess and improve the analytical methods used to predict the flow and heat transfer in high-temperature gas turbines. At the time the HOST project was initiated, an across-the-board improvement in turbine design technology was needed. A building-block approach was utilized and the research ranged from the study of fundamental phenomena and modeling to experiments in simulated real engine environments. Experimental research accounted for approximately 75 percent of the funding while the analytical efforts were approximately 25 percent. A healthy government/industry/university partnership, with industry providing almost half of the research, was created to advance the turbine heat transfer design technology base.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_t$</td>
<td>gas stream pressure</td>
</tr>
<tr>
<td>$R$</td>
<td>radius</td>
</tr>
<tr>
<td>$Re_L$</td>
<td>stator exit Reynolds number</td>
</tr>
<tr>
<td>$Re_x$</td>
<td>local Reynolds number</td>
</tr>
<tr>
<td>$T_C$</td>
<td>coolant temperature</td>
</tr>
<tr>
<td>$T_G$</td>
<td>gas stream temperature</td>
</tr>
<tr>
<td>$T_W$</td>
<td>airfoil temperature</td>
</tr>
<tr>
<td>$T_u$</td>
<td>turbulence intensity</td>
</tr>
<tr>
<td>$S$</td>
<td>surface distance</td>
</tr>
<tr>
<td>$St$</td>
<td>Stanton number</td>
</tr>
<tr>
<td>$St_0$</td>
<td>reference Stanton number</td>
</tr>
<tr>
<td>$U$</td>
<td>gas stream velocity</td>
</tr>
<tr>
<td>$U_m$</td>
<td>rotor wheel speed</td>
</tr>
<tr>
<td>$U_w$</td>
<td>moving surface velocity</td>
</tr>
<tr>
<td>$u'$</td>
<td>instantaneous velocity</td>
</tr>
<tr>
<td>$V$</td>
<td>coolant velocity</td>
</tr>
<tr>
<td>$W$</td>
<td>blade tip cavity width</td>
</tr>
<tr>
<td>$X$</td>
<td>axial length</td>
</tr>
<tr>
<td>$\Theta$</td>
<td>angle between mean velocity and major tip cavity axis</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>length scale</td>
</tr>
<tr>
<td>$\Omega$</td>
<td>rotational velocity</td>
</tr>
</tbody>
</table>
INTRODUCTION

Improved performance of aircraft gas turbine engines is typically accompanied by increased cycle pressure ratio and combustor exit gas temperature. The hot-section components of these turbojet/turbofan engines are subjected to severe aerothermal loads during the mission flight profile. Meeting the design goals of high cycle efficiency, increased durability of the hot-section components, and lower operating costs requires a multidisciplinary approach. Turbine Heat Transfer was one of the six disciplines addressed in the multidisciplinary Hot Section Technology (HOST) Project.

When the HOST Project was originally being planned, Stepka (1980), one of the originators of the project, performed an uncertainty analysis on the ability to predict turbine airflow temperatures. He estimated that the then current ability to predict metal temperature in an operating engine was within 100 K and that by testing prototypes this could be refined to within 50 K. He also suggested that the uncertainty in heat flux was on the external or the hot gas side surface of the airfoil was a principle contributor to the inability to predict metal temperatures; however, both internal and external surface heat transfer were important. Thus a level of uncertainty in external heat transfer contributed to an order of magnitude uncertainty in component life.

A typical cooled aircraft gas turbine blade is illustrated in Fig. 1, showing the intricate internal flow passages and the variety of heat transfer mechanisms at work. These include: impingement cooling, serpentine passages with turbulator surfaces, and pin fins, all in very short distance entrance lengths. These factors and subject to strong rotational forces. In addition, since most blades are film cooled, the internal mass balance is a variable. The complexity of the external flow field over the turbine blade is illustrated in Fig. 2. Heat transfer in the external flow field is characterized by: high Reynolds number forced convection with rotation, high free-stream turbulence, strong pressure and temperature gradients, surface curvature, and an unsteady flow field. In addition, and most important, the internal and external surface heat transfer coefficients are coupled through the metal walls. In fact, the turbine airfoil is a very compact, very complex, very efficient heat exchanger. This feature is particularly important in a durability program, such as high cycle fatigue, and we Physics Fluids, Fluids at high Reynolds number in a rotating turbine. Another experiment acquired data on the internal turbulated serpentine passages subject to rotation at engine condition levels. Finally, vane heat transfer data were acquired in real engine type environment behind an actual operating combustor.

Over the life of the HOST Project a little less than $5 million net research dollars were invested in the Turbine Heat Transfer Subproject. As shown in Fig. 4(a), there was a healthy government/industry/university partnership with industry providing almost half the research effort. Approximately three-fourths of the effort was experimental, as shown in Fig. 4(b). The majority of effort established a large number of major experimental datasets. These datasets have been well received and expected to provide benchmarks for turbine heat transfer for many years to come. The analyses covered a wide range, including a three-dimensional Navier-Stokes effort; however, most of the analytic effort was focused on modeling local phenomena of key importance. The scope and range of the program is best seen by examining representative results.

EXPERIMENTAL DATABASE

The experimental part of the Turbine Heat Transfer Subproject consisted of 21 large experiments and three (3) of somewhat more modest scope and was structured to address the phenomena identified in Figs. 1 and 2. Three (3) of the large experiments were conducted in a stationary frame of reference and three (3) were conducted in a rotating frame of reference.

Stationary Reference

One of the initial research efforts was the stator airfoil heat transfer program performed at the Allison Gas Turbine Division (Nealy et al., 1983; Hylton et al., 1983; Nealy et al., 1984; Turner et al., 1985; Yang et al., 1985). This research consisted of determining the effects of Reynolds number, turbulence level, Mach...
number, temperature ratio, acceleration, and boundary layer transition on heat transfer coefficients for various airfoil geometries at simulated engine conditions. This research was conducted for nonfilm-cooled airfoils, showerhead film-cooled designs and showerhead/gill-region film cooling concepts. Typical results of this research are shown in Fig. 5. A typical cascade configuration is shown in the photograph (Fig. 5(a)). Two-dimensional midspan heat transfer coefficients and static pressure distributions were measured on the central airfoil of the three vane cascade. Nonfilm-cooled data are shown in Fig. 5(b) where the boundary layer transition is clearly identified as a function of Reynolds number on the suction surface. Figure 5(c) shows the effect on heat transfer in the downstream recovery region to the addition of showerhead film cooling. Data are presented as a Stanton number reduction. A detrimental effect is noted in the boundary layer transition region of the suction surface to the addition mass at the leading edge. Figure 5(d) shows a strong dependence on “gill-region” film cooling which is consistent with experience. However, when combining showerhead with all-region film cooling more mass addition is not always better as indicated by the Stanton number reduction data on the pressure surface. This is a very extensive database which systematically shows the importance of modern film cooling schemes on modern airfoils. It went beyond the traditional effectiveness correlations to provide actual heat transfer data. It should provide a valuable baseline for emerging analysis codes.

An investigation of secondary flow phenomena in a 90° curved duct was conducted at the University of Tennessee Space Institute (Crawford et al., 1985). The curved duct was utilized to represent airfoil passage curvature without the complexity of the horseshoe vortex. These data consist of simultaneous three-dimensional mean value and fluctuating components of velocity through the duct. The research results are presented in the literature. A schematic of the test facility and the three-dimensional laser velocimeter are shown in Fig. 6. The first phase of the research examined flows with a relatively thin inlet boundary layer and low free-stream turbulence. The second phase studied a thicker inlet boundary layer and higher free-stream turbulence. Typical experimental results of this research are summarized in Fig. 6. The vector plot of cross-flow velocities clearly shows the development of a vortex in the duct corner near the low pressure surface. The analytical results will be mentioned in the Viscous Flow Analysis section. These data provide a comprehensive benchmark to verify codes at realistic flow conditions. Two experiments were also conducted at NASA Lewis in the high-pressure facility (Gladden et al., 1985a; Gladden et al., 1985b; Gladden et al., 1987; Htppensteele et al., 1985). This facility was capable of testing a full-sized single-stage turbine at simulated real engine conditions. The tests, however, were limited to combined combustor/stator experiments. One experiment examined full-coverage film-cooled stator airfoils, while the second experiment utilized some of the advanced instrumentation developed under the instrumentation subproject. A comparison of experimental airfoil temperatures with temperatures obtained from a typical design system showed substantial differences for the full-coverage, film-cooled airfoils and suggests that models derived from low-temperature experiments are inadequate for “real-engine” conditions. The advanced instrumentation tests demonstrated the capability and the challenges of measuring heat flux and time-resolved gas temperature fluctuation in a real-engine environment.

Typical results are shown in Fig. 7 for thin film thermocouples and the dynamic gas temperature probe tested a simulated real engine condition. A comparison is made between steady state heat flux measurements and those determined from dynamic signal analysis techniques.

Stanford University has conducted a systematic study of the physical phenomena that affect heat transfer in turbine airfoil passages. Their recent experimental research has been concerned with high free-stream turbulence intensity and large turbulence scale that might be representative of combustor exit phenomena. A schematic of their free jet test facility and typical results are shown in Fig. 8. Data are measured on a constant temperature flat plate located at a specified radial and axial distance from the jet exit centerline. These data, presented as Stanton number ratios, indicate that heat transfer augmentation can be as high as 5X at a high value of free-stream turbulence intensity but only 3X if the length scale is changed. These results suggest that the designer must know a great deal more about the aerodynamic behavior of the flow field in order to fully exploit the thermal performance of the turbine components.

Prior to the advent of the HOST program, Arizona State University was pursuing a systematic study of impingement heat transfer with cross-flow characteristic of turbine airfoil cooling schemes. The work was initially sponsored by a NASA Lewis grant but was subsequently funded by the HOST program. The results of this research are summarized in Florschuetz et al. (1982a), Florschuetz et al. (1982b), Florschuetz et al. (1982c), Florschuetz et al. (1983), Richards et al. (1984), Florschuetz et al. (1984), Florschuetz et al. (1985), Florschuetz et al. (1984). In addition to the many geometry variations, this research also investigated the effects of various jet-flow to crossflow ratios and differences between the jet-flow and the cross-flow temperature. Correlations were developed for both inline rows of impingement jets and staggered arrays of jets but without an initial cross-flow. The effects of cross-flow and temperature differences were then determined relative to the base correlations.

Rotating Reference

In the rotating reference frame, experimental aerodynamic and heat transfer measurements were made in the large, low-speed turbine at the United Technologies Research Center (Dring et al., 1987; Dring et al., 1986a; Dring et al., 1986b; Dring et al., 1986c; Blair et al., 1988; Blair et al., 1988). Single-stage data with both high and low-inlet turbulence were taken in phase I. The second phase examined a one and one-half stage turbine and focused on the second vane row. Under phase III aerodynamic quantifies such as interrow time-averaged and rms values of velocity, flow angle, inlet turbulence, and surface pressure distributions were measured. A photograph of the test facility is shown in Fig. 9. Typical heat transfer data for both the first stator and rotor are shown in Fig. 10. The data show that an increase of inlet turbulence has a substantial impact on the first stator heat transfer. However, the impact on the rotor heat transfer is minimal. These data are also compared with Stanton numbers calculated by a boundary layer code and the assumption that the boundary layer was either laminar (LAM) or fully turbulent (FULT).

These assumptions are made in the data on the suction surface of both the stator and the rotor. However, the heat transfer on the pressure surface, especially for the high turbulence case, was generally above even fully turbulent levels on both airfoils. Pressure surfaces have traditionally received less...
The blade tip region of rotor blades is often a critical region and an area that suffers substantial damage from the high temperature environment. Arizona State University has experimentally modeled the blade tip cavity region and determined heat transfer rates by a mass transfer analogy with naphthylene (Chyu et al., 1987). A schematic of the test is shown in Fig. 10. The blade tip cavity is a stationary model and the relative velocity of the shroud is represented by a moving surface at a specified “gap” spacing from the blade. Stanton number results for two different cavity aspect ratios are also shown in Fig. 10. The heat transfer on the surfaces next to the shroud are little changed by the aspect ratio which is not surprising. However, the heat transfer to the floor of the cavity is increased significantly on the downstream portion at the lower aspect ratio. Also shown in the figure is the flow angle effect on heat transfer. Because of the airfoil turning at the tip the cavity will be at different angles of attack to the mean crossflow direction. The data shows a minimal effect at an aspect ratio of 0.9 and a substantial effect at an aspect ratio of 0.23. This dataset is really quite a new addition to a tradition of neglect area and shows that with careful datasets and analyses one can obtain an optimal design for tip cavities.

The preceding studies were focused on the hot-gas side phenomena. Since the heat transfer phenomena is driven by the hot-gas side conditions, it is appropriate to concentrate resources on this area. However, the coolant-side heat transfer is also important. Therefore, coolant passage heat-transfer and flow measurements in a rotating reference frame were also obtained at Pratt & Whitney Aircraft/United Technologies Research Center (Kopper, 1984; Sturgess et al., 1987; Lord et al., 1987). Experimental data were obtained for smooth-wall serpentine passages and for serpentine passages with skewed and normal turbulators. The flow and rotation conditions were typical of those found in actual engines. This was a very realistic experiment. Data for both the smooth-wall and skewed turbulator passages are shown in Fig. 11 for radial outflow, representing only a tiny fraction of the total data involved in this very complex flow. Both datasets are shown correlated with the rotation number except for high rotation numbers where high pressure surface is an area that requires additional research to understand and model the physical phenomena occurring in these passages.

**ANALYTICAL TOOLS**

The analytic parts of the turbine heat transfer subproject are characterized by efforts to adapt existing codes and analyses to turbine heat transfer. In general, these codes and analyses were well established before HOST became involved; however, the applications were not for turbine heat transfer, and extensive revision has often been required. In some cases the analytic and experimental work were part of the same contract.

**Boundary Layer Analysis**

The STANS boundary-layer code (Crawford et al., 1976) (which was developed on NASA contract at Stanford University in the mid-1970's) was modified by Allison Gas Turbine Division to define starting points and transition length of turbulent flow to accommodate their data, with and without film cooling, as well as data in the literature. Specific recommendations are made to improve turbine airfoil heat transfer modeling utilizing a boundary layer analysis. These recommendations address the boundary conditions, the initial condition specification, including both velocity and thermal profiles, and modifications of conventional zero order turbulence models. The results of these improvements are shown in Fig. 12 where the start of transition and its extent on the suction surface are reasonably well characterized. For the case of showerhead film cooling, two empirical coefficients were utilized: one to modify the gas stream enthalpy boundary conditions and the gas stream enthalphy boundary conditions and permit a representative prediction of the Stanton number reduction in the recovery region. Boundary layer methods can be used for midspan analysis, however they require a realistic data base to provide the coefficients needed for proper reference.

In another boundary layer code effort United Technologies Research Center assessed the applicability of its three-dimensional boundary layer code to calculate heat transfer total pressure loss and streamline flow patterns in turbine passages. The results indicate a strong three-dimensional effect on a turbine blade, and agrees qualitatively with experimental data. The same code was modified for use as a two-dimensional unsteady code in order to analyze the rotor-stator interaction phenomenon (Vatine, 1983; Anderson et al., 1985b; Anderson, 1985c). These codes also needed data as input.

Finally, a fundamental study on numerical turbulence modeling, directed specifically at the airfoil in the turbine environment, was conducted at the University of Minnesota. A modified form of the Lam-Bremhorst low-Reynolds-number k-ε turbulence model was developed to predict transitional boundary layer flows under conditions characteristic of gas turbine blades (Schmidt et al., 1987) including both free-stream turbulence and pressure gradient. The purpose was to extend previous work on turbulence modeling to apply the model to transitional flows with both free-stream turbulence and pressure gradients. The results of the effort are compared with the experimental data of Allison Gas Turbine Division in Fig. 13. The augmentation of heat transfer on the pressure surface over the fully turbulent value is predicted reasonably well. In addition, when an adverse pressure gradient correction is utilized, the suction surface heat transfer data is also predicted reasonably well.

This was a reasonably good beginning to establishing a methodology for moving away from the heavy dependence on empirical constants. Although boundary layer methods will never solve the whole problem, they will always remain important analytic tools.

**Viscous Flow Analysis**

The three-dimensional Navier-Stokes TEACH code has been modified by Pratt & Whitney for application to internal passages and to incorporate rotational terms. The modified code has been delivered to NASA Lewis and tested on some simple geometric cases. The results of this effort indicate that the code is qualitatively adequate for simple geometric cases of practical interest, much work remains to be done to bring the internal passage computational codes up to the level of proficiency of the free-stream codes. For the external airfoil surface important analytic progress is being made. By contrast, the work on internal passages is still primitive. The internal problem is substantially more complex.

A fully elliptic three-dimensional Navier-Stokes code has been under development at Scientific Research Center.

**ORIGINAL PAGE IS OF POOR QUALITY**
Assessments (SRA) for many years. This code was primarily directed at inlets and nozzles. SRA, Inc., has modified the code for turbine applications (Weinberg et al., 1985). This includes grid work for turbine airfoils, adding an energy equation and turbulence modeling, improving the inlet and rotor-stator of a stage and one-half axial turbine was calculated for the curved duct experiment utilizing the P.D. Thomas code (Thomas, 1979) as a base. Some analytical results from this code are shown in Fig. 6 where a vector plot of the cross-flow velocities are compared with the experiment. In addition, a stream sheet is shown as it propagates through the duct and is twisted and stretched. Additional comparisons of analyses and experiment show that the thin turbulent boundary layer results of this experiment are difficult to calculate with current turbulence models.

CONCLUDING REMARKS

Since this paper is an overview of the Turbine Heat Transfer aspects of the HOST program it has been presented as a cataloging and summarizing of the various activities. More importantly, the HOST program should be viewed as a catalyst bringing together the gas turbine community and building a technology momentum to carry advanced propulsion systems into the future. Specifically, the HOST Turbine Heat Transfer Subproject can point to the following accomplishments:

1. The impact of axial spacing and inlet turbulence on heat transfer and aerodynamics throughout the stator-rotor-stator of a stage and one-half axial turbine was measured. High-turbulence and post-transitional effects on boundary layer transition and heat transfer were determined for a stator airfoil.
2. Reynolds number, Mach number, curvature, and wall-to-gas temperature effects on boundary layer transition and heat transfer were determined for a stator airfoil.
3. Showerhead and "gill-region" film-cooling were shown to have both beneficial and adverse effects on the recovery region heat transfer at simulated engine conditions which depend on specific operating conditions.
4. Heat Transfer in both smooth-wall and turbulent-wall serpentine rotating coolant passages were correlated with a rotation number for the low-pressure surface. The high-pressure surface heat transfer was not well correlated.
5. Blade tip cavity heat transfer was shown to be strongly dependent on the cavity aspect ratio and angle-of-attack to blade tip flow direction.
6. Heat transfer measurements in high-turbulence intensity flow fields, simulating combustor exit phenomena, shows augmentation rates of 3X to 5X depending on the length scale of the turbulence.
7. Improved definition of the initial conditions and boundary conditions which are applicable to turbine airfoils was successful in improving the prediction of airfoil heat transfer for a wide range of geometries using the STANS boundary layer code.
8. The Lamin-Breuerst low-Reynolds number k-ε turbulence model was modified to also improve the predictions of airfoil heat transfer under transitional flows with both free-stream turbulence and pressure gradients.
9. A fully elliptic Navier-Stokes code was developed for turbine airfoils and includes turbulence modeling, an energy equation and improved user friendliness. Frequently, heat transfer is a limiting factor in the performance and durability of an engine. However, as new continuous-time, lower speed, advancing technology becomes an interdisciplinary effort involving aerothermal loads definition and the structural response of advanced materials. This is especially true for hypersonic vehicles where the overall thermal management and design of the vehicle and the propulsion system become an integrated interactive entity. We now have, or are developing, tremendous analytical capabilities with which one can attack these very complex technology issues.

LOOK TO THE FUTURE

Many recent studies have been made to assess the aeropropulsion technology requirements into the 21st century. The consensus seems to suggest that significant technology advances are required to meet the goals of the future. Whether the goals are high speed sustained flight, single-stage-to-orbit or subsonic transport, the issues for the designer are improved fuel efficiency, high thrust-to-weight, improved component performance while maintaining component durability and reduced operating and maintenance costs. These issues will only serve to increase the "opportunities" available to the researcher in aerothermal loads and structures analysis. The verifiable predictions of unsteady flowfields with significant secondary flow phenomena and coupled thermal/velocity profiles are a fertile research area. Very little progress has been made to date in applying CFD techniques to the intricate and complex coolant channel required in the hot-section components. With the expected advances in high-temperature materials the components with significant aerothermal loads problems will expand beyond the airfoils and combustor liners to shrouds, rims, seals, bearings, compressor blading, ducting, nozzles, etc. The issues to be addressed and the technology advances required to provide the aeropropulsion systems of the 21st century are quite challenging.

REFERENCES


### TABLE I. - TURBINE HEAT TRANSFER SUBPROJECT SUMMARY

<table>
<thead>
<tr>
<th>Figures 1 and 2</th>
<th>Work element</th>
<th>Results</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nonrotating experiments</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>Airfoil with film cooling</td>
<td>Provide fundamental experimental data bases with focus on -</td>
<td>2 to 6</td>
</tr>
<tr>
<td>11</td>
<td>Curved duct</td>
<td>Film cooling</td>
<td>7</td>
</tr>
<tr>
<td>3</td>
<td>Impingement cooling</td>
<td>Secondary flows</td>
<td>12 to 19</td>
</tr>
<tr>
<td>5</td>
<td>Large-scale, high-intensity turbulence</td>
<td>Impingement pattern correlations</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>Real engine environment</td>
<td>Combustor exit simulation</td>
<td>8 to 10</td>
</tr>
<tr>
<td>Rotating experiments</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>Large low-speed turbine</td>
<td>Rotor-stator interaction</td>
<td>20 to 25</td>
</tr>
<tr>
<td>2</td>
<td>Rotating coolant passage</td>
<td>Coriolis and buoyancy effects</td>
<td>27</td>
</tr>
<tr>
<td>4</td>
<td>Tip region simulator</td>
<td>Flow across moving airfoil tip</td>
<td>26</td>
</tr>
<tr>
<td>9</td>
<td>Warm core turbine</td>
<td>Vane and blade passage flow map fully scaled</td>
<td></td>
</tr>
<tr>
<td>Analyses</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>STAN5 modifications</td>
<td>Enhance analytic tools for turbine application</td>
<td>2 to 5</td>
</tr>
<tr>
<td>10</td>
<td>Three-dimensional boundary layer</td>
<td>Adapt boundary layer code to current airfoil data</td>
<td>30 to 34</td>
</tr>
<tr>
<td>8</td>
<td>Unsteady boundary layer</td>
<td>Zoom focus on Three-dimensional regions</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Teach code with rotation</td>
<td>Account for rotor-stator interaction effects</td>
<td>28 to 29</td>
</tr>
<tr>
<td>7</td>
<td>Low Reynolds number</td>
<td>Three-dimensional Navier-Stokes with rotation terms</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>Mint code</td>
<td>Develop turbine airfoil specific turbulence model</td>
<td>35</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Three-dimensional Navier-Stokes applied to turbine airfoil geometry</td>
<td>36</td>
</tr>
</tbody>
</table>

*aExperiment and analysis in the same contract.

*bWork done under two separate contracts.
FIGURE 1. - TYPICAL COOLED AIRCRAFT GAS TURBINE BLADE. SEE TABLE I FOR DESCRIPTION OF NUMBERED FLOW PHENOMENA.

HIGHLY TURBULENT REAL ENGINE FLOW SIMULATION

FIGURE 2. - COMPLEX FLOW PHENOMENA IN A TURBINE PASSAGE. SEE TABLE I FOR DESCRIPTION OF NUMBERED FLOW PHENOMENA.
FIGURE 3.- BUILDING BLOCK APPROACH TO TURBINE AEROTHERMAL RESEARCH.

CD-85-16940
FIGURE 4.- HOST TURBINE HEAT TRANSFER SUBPROJECT DISTRIBUTION OF RESOURCES.
FIGURE 5. - GAS-SIDE EXPERIMENTAL HEAT TRANSFER DATA FOR BOTH NON-FILM-COOLED AND FILM COOLED AIRFOILS. ALLISON GAS TURBINE DIVISION.
Figure 6. - Three-dimensional flow-field measurements in a curved duct representing an airfoil passage. University of Tennessee Space Institute.
WALL TEMPERATURE RESPONSE TO GAS TEMPERATURE RAMP

LOCAL HEAT TRANSFER COEFFICIENTS DETERMINE FROM DYNAMIC SIGNAL ANALYSIS

CD-85-17458

FIGURE 7. - HEAT FLUX MEASUREMENTS MADE IN A SIMULATED REAL ENGINE ENVIRONMENT ON STATOR AIRFOILS.
LEWIS RESEARCH CENTER.
Figure 8. - Heat transfer augmentation resulting from high free stream turbulence and scale.
Stanford University.
LARGE SCALE ROTATING RIG 1-1/2 STAGE TURBINE CONFIGURATION FIRST VANE AND ROTOR CASE REMOVED

FIGURE 9. - ROTOR/STATOR INTERACTION AND THE AFFECT ON HEAT TRANSFER OF HIGH AND LOW FREESTREAM TURBULENCE. UNITED TECHNOLOGIES RESEARCH CENTER.
FIGURE 10. - EXPERIMENTAL HEAT TRANSFER RESULTS FOR A SIMULATED BLADE TIP CAVITY. ARIZONA STATE UNIVERSITY.
FIGURE 11. - THE EFFECTS OF ROTATION ON HEAT TRANSFER IN MULTIPASS COOLANT PASSAGES WITH AND WITHOUT TURBULATORS ARE SHOWN FOR AN OUTWARD FLOWING PASSAGE. PRATT AND WHITNEY AIRCRAFT.
FIGURE 12. - A MODIFIED STAN-5 BOUNDARY LAYER ANALYSIS IS COMPARED WITH MEASUREMENTS FROM A NON-FILM-COOLED AIRFOIL AND THE SAME AIRFOIL GEOMETRY WITH A SHOWER-HEAD DESIGN. ALLISON GAS TURBINE DIVISION.
TURBULENCE = 6.5 PERCENT; EXIT MACH NUMBER = 0.90

FIGURE 13. - A MODIFIED LOW REYNOLDS NUMBER K-ε TURBULENCE MODEL FOR A BOUNDARY LAYER ANALYSIS IS COMPARED WITH EXPERIMENTAL MEASUREMENTS. UNIVERSITY OF MINNESOTA.
FIGURE 14. - A FULLY ELLIPTIC NAVIER-STOKES SOLUTION IS COMPARED WITH THE SAME EXPERIMENTAL MEASUREMENTS FOUND IN FIGURES 12 AND 13. SCIENTIFIC RESEARCH ASSOCIATE, INC.
HOST Turbine Heat Transfer Program Summary

Herbert J. Gladden and Robert J. Simoneau

National Aeronautics and Space Administration Lewis Research Center
Cleveland, Ohio 44135-3191

Sponsoring Agency Name and Address
National Aeronautics and Space Administration
Washington, D.C. 20546-0001


The objectives of the HOST Turbine Heat Transfer subproject were to obtain a better understanding of the physics of the aerothermodynamic phenomena and to assess and improve the analytical methods used to predict the flow and heat transfer in high temperature gas turbines. At the time the HOST project was initiated, an across-the-board improvement in turbine design technology was needed. A building-block approach was utilized and the research ranged from the study of fundamental phenomena and modeling to experiments in simulated real engine environments. Experimental research accounted for approximately 75 percent of the funding while the analytical efforts were approximately 25 percent. A healthy government/industry/university partnership, with industry providing almost half of the research, was created to advance the turbine heat transfer design technology base.

Heat transfer
Gas turbines
Film cooling

Unclassified - Unlimited
Subject Category 34