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>Definition</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_2</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>Tu</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_o</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>@</td>
<td>angle between mean velocity and major tip cavity axis</td>
</tr>
<tr>
<td>@</td>
<td>length scale</td>
</tr>
<tr>
<td>@</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 airfoil 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 principal contributor to the inability to predict metal temperatures; however, both internal and external surface heat transfer were important. The levels of uncertainty in thermal performance 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 (i.e., entrance length) distances 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, and very efficient heat exchanger. This feature is particularly important in a durability program, such as improved turbine heat transfer points on the turbine airfoil. Film cooled airfoils, passage curvature, endwall flows, transitioning blade boundary layers, tip regions, and free-stream turbulence on the external surfaces. The subproject included impingement and turbulated serpentine passages on the internal surfaces. The program broke some new ground. An experiment was conducted, which obtained heat transfer data on the surfaces of the airfoils in a one and one-half stage large low speed 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 a 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. 1, 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. 2. The majority of effort established a large number of major experimental datasets. These datasets have been with 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 six (6) 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 airflow 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 film-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 dataset which systematically shows the important effects 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 airflow 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 and compliment similar data 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 crossing flow 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 shown 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 used a modified version of the advanced instrumentation developed under the instrumentation subproject. The performance of experimental airfoil's temperature 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 in "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. They are 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 3X 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 successfully predict 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. (1983), Richards et al. (1984), Florschuetz et al. (1984), Florschuetz et al. (1985), Florschuetz et al. (1987). 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 line 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). Typical data are presented as Stanton number ratios, which are the ratio of the measured Stanton number to the Stantons number obtained from a detailed calculation of the thermal performance of the turbine components. 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 successfully predict 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. (1983), Richards et al. (1984), Florschuetz et al. (1984), Florschuetz et al. (1985), Florschuetz et al. (1987). 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 line 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). Typical data are presented as Stanton number ratios, which are the ratio of the measured Stanton number to the Stantons number obtained from a detailed calculation of the thermal performance of the turbine components. 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 successfully predict 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. (1983), Richards et al. (1984), Florschuetz et al. (1984), Florschuetz et al. (1985), Florschuetz et al. (1987). 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 line 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). Typical data are presented as Stanton number ratios, which are the ratio of the measured Stanton number to the Stantons number obtained from a detailed calculation of the thermal performance of the turbine components. 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 successfully predict 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. (1983), Richards et al. (1984), Florschuetz et al. (1984), Florschuetz et al. (1985), Florschuetz et al. (1987). 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 line 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.
attention than suction surfaces. The high heat transfer on the pressure surface is not readily explainable and calls for additional research, especially modeling, on pressure surfaces.

The 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 napthylene (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 neglected area and shows that with careful data collection and analysis one can obtain an optimal design for tip cavities.

The preceding studies were concerned with 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 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 the pressure side 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 to modify the wall temperature and the gas-phase enthalpy 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 three-dimensional effect on a turbine blade, and agree qualitatively with experimental data. The code was modified for use as a two-dimensional unsteady code in order to analyze the rotor-stator interaction phenomena (Vatn, 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-e 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 establish 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, and 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
on the pressure surface of both stator and rotor can
be viewed as a catalyst bringing together the gas tur-
bine community and building a technology momentum to
carry advanced propulsion systems into the future.
Future propulsion systems will require increased en-
ergy conversion potential, reduced emissions, and
turbulent flow. To meet these requirements, impor-
tant 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
compound 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 aero-
thermo loads and structures analysis. The verifiable
predictions of unsteady flowfields with significant
secondary flow phenomena and coupled thermal/velocity
profiles is a fertile research area. Very little pro-
gress has been made to date in applying CFD techniques
to the intricate and complex coolant channels 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 tech-
ology advances required to provide the aeropropulsion
systems of the 21st century are quite challenging.

CONCLUDING REMARKS

Since this paper is an overview of the Turbine Heat Transfer as-
pects of the HOST program it has been pre-
ented as a cataloging and summarizing of the various
activities. More importantly, the HOST program should
be viewed as a catalyst bringing together the gas tur-
bine 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 the pressure surface. The high-pressure surface 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
phena, shows augmentation rates of 3X to 5X depend-
ing on the length scale of the turbulence.
7. Improved definition of the initial conditions
and boundary conditions which are applicable to tur-
bine 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 Lam-Bremhorst low-Reynolds number k-ε tur-
bulence model was modified to also improve the predic-
tions of airfoil heat transfer under transitional flows
with both free-stream turbulence and pressure gradients.

REFERENCES

Anderson, O. L., 1985a, "Assessment of a 3-D Boundary
Layer Analysis to Predict Heat Transfer and Flow Field
Anderson, O. L., 1985b, "Calculation of Three-
Dimensional Boundary Layers on Rotating Turbine
Blades," Three Dimensional Flow Phenomena in Fluid
Machinery, A. Hamed, J. Herring, and L. Povinelli, ed.,
ASME, New York, pp. 121-132.
Blair, M. F., Dring, R. P., and Joslyn, H. D., 1988a,
"The Effects of Turbulence and Stator/Rotor Interactions
on Turbine Heat Transfer, Part II - Effects of Reynolds
Number and Incidence," ASME Paper to be presented at
the 33rd ASME Gas Turbine Conference, The Netherlands.
Blair, M. F., Dring, R. P., and Joslyn, H. D., 1988b,
"The Effects of Turbulence and Stator/Rotor Interactions
on Turbine Heat Transfer, Part I - Design Operating
Conditions," ASME Paper to be presented at the 33rd ASME
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(^a)</td>
<td>Provide fundamental experimental data bases with focus on - film cooling</td>
<td>2 to 6</td>
</tr>
<tr>
<td>11</td>
<td>Curved duct</td>
<td>Secondary flows</td>
<td>7</td>
</tr>
<tr>
<td>3</td>
<td>Impingement cooling</td>
<td>Impingement pattern correlations</td>
<td>12 to 19</td>
</tr>
<tr>
<td>5</td>
<td>Large-scale, high-intensity turbulence</td>
<td>Combustor exit simulation</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>Real engine environment</td>
<td>The real environment</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(^a)</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>STANS modifications(^a)</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></td>
</tr>
<tr>
<td>8</td>
<td>Unsteady boundary layer(^a)</td>
<td>Zoom focus on Three-dimensional regions</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Teach code with rotation(^a)</td>
<td>Account for rotor-stator interaction effects</td>
<td></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(^b)</td>
<td>Develop turbine airfoil specific turbulence model</td>
<td></td>
</tr>
</tbody>
</table>

\(^a\)Experiment and analysis in the same contract.
\(^b\)Work done under two separate contracts.
FIGURE 1. - TYPICAL COOLED AIRCRAFT GAS TURBINE BLADE. SEE TABLE I FOR DESCRIPTION OF NUMBERED FLOW PHENOMENA.

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.
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.
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 showerhead design. Allison Gas Turbine Division.
TURBULENCE = 6.5 PERCENT; EXIT MACH NUMBER = 0.90

FIGURE 13. - A MODIFIED LOW REYNOLDS NUMBER K-Ɛ TURBULENCE MODEL FOR A BOUNDARY LAYER ANALYSIS IS COMPARED WITH EXPERIMENTAL MEASUREMENTS. UNIVERSITY OF MINNESOTA.

CD-87-29102
TYPICAL O-GRID MESH

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.
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.