

NASA Technical Memorandum 102460

Recent Progress in Research Pertaining to Estimates of Gas-Side Heat Transfer in an Aircraft Gas Turbine

Robert W. Graham
Lewis Research Center
Cleveland, Ohio

Prepared for the
1990 Turbo Expo
sponsored by the American Society of Mechanical Engineers
Brussels, Belgium, June 11-14, 1990



(NASA-TM-102460) RECENT PROGRESS IN
RESEARCH PERTAINING TO ESTIMATES OF GAS-SIDE
HEAT TRANSFER IN AN AIRCRAFT GAS TURBINE
(NASA) *12*

CSCL 21F

N70-13394

858280

unclas

G3/07 0252614

RECENT PROGRESS IN RESEARCH PERTAINING TO
ESTIMATES OF GAS-SIDE HEAT TRANSFER IN AN
AIRCRAFT GAS TURBINE

Robert W. Graham
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio

ABSTRACT

A decade ago several important fundamental heat transfer phenomena were identified which were considered basic to the ability to predict heat transfer loads in aircraft gas turbines. The progress in addressing these fundamentals over the past ten years is assessed in this paper. Much research effort has been devoted to their study in university, industry and government laboratories and significant progress has been achieved. Advances in computer technology have enabled the modeling of complex three-D fluid flow in gas turbines so necessary for heat transfer calculations. Advances in instrumentation plus improved data acquisition have brought about more reliable data sets. While much has advanced in the '80's, much challenging research remains to be done. Several of these areas are suggested in the paper.

INTRODUCTION

Approximately 10 years ago I wrote an ASME paper entitled "Fundamental Mechanisms That Influence the Estimate of Heat Transfer to Gas Turbine Blades" (1). The intent of the paper was to highlight a number of fundamental physical phenomena that are considered to be significant influences on the gas-side heat transfer to a turbine blade. It was suggested that the generic nature of the phenomena would entice researchers both within and outside the gas turbine technical community to initiate further research investigations which addressed some of the fundamental problems.

The intriguing possible application of a class of fundamental research programs in advanced turbine design did seem to encourage research activity. I don't mean to infer that my paper was the principal inciter of that activity across the research community. It was influential in guiding the planning of the rejuvenated research program in gas turbine heat transfer at the Lewis Research Center in the early '80's. Over the past decade there has been a flurry of research activity done cooperatively, or independently, by universities, industry and government

laboratories. One of the best known cooperative ventures was the HOST program sponsored by NASA and managed by the Lewis Research Center. This program addressed an array of technologies important to the aviation gas turbine engine and included a sizeable effort in turbine heat transfer. Department of Defense initiatives sponsored by the Air Force, Army, and the Navy also gave strong impetus to heat transfer research programs conducted in industry, universities, and government laboratories. The Integrated High Performance Turbine Engine Technology (IHPTET) is an example of a coordinated program aimed at advancing turbine engine technology. The Independent Research and Development (IR&D) plans of the aircraft engine industry reflected major programs in fundamental and applied turbine heat transfer research.

In view of the progress that has been made over the past decade, I thought it would be interesting and productive to reassess the state of the research in the areas that were suggested in my original paper (1). The principal research areas suggested in the original paper were the following:

- a) Leading edge or stagnation heat transfer where unsteady components of flow influence the hydrodynamic and thermal boundary layers.
- b) The influence of passage geometry (primarily curvature and local conditions of acceleration or deceleration) on the blade passage boundary layers.
- c) The influence of secondary flows promoted by pressure gradients, end wall conditions, body forces, and local vortices on blade boundary layers. These induce a complex three-D flow in the turbine passages.
- d) The general unsteady nature of the flow entering the turbine with wakes introduced in the compressor and in the combustor and also temperature streaks and patterns that add a thermal unsteadiness to the gases. The thermal and hydrodynamic unsteadiness is thought to influence the

development of the boundary layers on the turbine blade surfaces. The question of the transition mechanism is complicated by the unsteady nature of the flow.

COMPUTATIONAL PREDICTIONS OF HEAT TRANSFER

Over the past decade, significant progress has been made in the prediction of gas-side heat transfer rates or heat transfer coefficients. Prior to this period, crude estimates of heat transfer were made with correlations. Such practice was replaced with boundary layer analysis; both integral and differential formulations. The STAN 5 boundary layer code developed at Stanford (2) is a specific example of a predictive code that has had wide usage. Researchers and designers continue to adopt it, or similar boundary layer analysis, in estimating local heat transfer rates along blade or stator surfaces. Specifying appropriate initial conditions and transition criteria are among the most difficult aspects of the approach.

The increased storage and processing capabilities of advanced computers have enabled two-D and three-D flow Navier-Stokes solvers to be developed. In their present state, these codes generally assume steady or quasi-steady flow conditions. Such fluid flow solutions are essential prerequisites to estimates of convective heat transfer where fluid velocity and fluid properties are the principal parameters. For example the MINT code (3) is a three-dimensional Navier-Stokes solver that predicts the flow field in a channel of a turbine. The three-dimensional capability of the Navier-Stokes solvers are essential in dealing with the complex flow geometrics of the blade rows. The intersection of the blades or vanes at the endwall or hub region cause a complex secondary flow vortex to develop. It is appropriately called a horseshoe vortex that straddles the leading edge of the blades or vanes. Dr. Roderick Chima of LeRC (4) has developed a Navier-Stokes solver for this complex region. Calculations have been made of the horseshoe vortex formed in front of a rounded leading edge in turbulent flow and compared to experimental data. The comparisons were favorable. Other comparisons have been made with data obtained in an annular turbine cascade. These also appeared to be favorable. The authors of reference 4 concluded that additional experience with the analysis is needed, but it shows promise as a useful tool for three-D viscous flow analysis in general. Perhaps some of these three-D Navier-Stokes solvers won't become subroutines in design procedure. However, they will be highly useful as tools to delineate critical regions and may serve as flow models for the more complex blade shapes that are being proposed for advanced turbines. These complex three-D flow analyses are being verified in carefully designed experiments that simulate the initial conditions and boundary conditions of the computational model. It is anticipated that the verification process will generate greater understanding of the flow physics of the phenomena. Such new understanding will contribute to the design guidelines for advanced turbine blades. Thus, highly sophisticated three-D codes describing complex flow phenomena such as the horseshoe vortices, will make a substantial contribution to the design of radically different blade shapes under consideration.

The authors of reference 5 have developed a

three-D viscous code applicable to linear cascades with flat endwalls and have compared its predictions in reference 6 with experimental heat transfer data derived from a large-scale cascade facility. The comparison between experiment and prediction looks favorable for both heat transfer and fluid friction. The comparisons were carried out over a limited range of experimental conditions. Nevertheless, the results do represent a significant achievement in computational prediction of heat transfer for a three-D geometry.

Advanced computational methods of the past decade are making their way into the design procedure for cooled turbine blades. In addition to prescribing the local heat transfer loads entering the blade or vane surfaces, these codes prescribe the cooling passage design and even the film cooling slots or holes to meet the desired blade metal temperatures. Further, they may include a structural analysis which utilizes the blade temperature distribution as input. There is a definite trend in the composition of these design codes to include features of expert systems which automatically make decisions from an array of options. These design codes which are a system of computational routines, include elaborate graphics packages which show the results in color-coded displays.

All of the major aircraft gas turbine industries are developing design systems similar to what was described above to varying degrees of sophistication. One example is a design procedure developed at General Electric called "Auto-Airfoil". It was described in reference 5. The method evolved from a joint effort between the Lynn, Massachusetts, Advanced Computer-Aided Engineering Group and the Evendale, Ohio, Heat Transfer Design Methods Group. It consists of four principal modules which include Cooling Geometry Design, Cooling Analysis, Boundary Condition Generator, and Detailed Thermal Analysis. The prescribed input to the system includes blade outer contour geometry, local gas Mach numbers and pressure distribution. The design module specifies the requirements for weight, strength, and cost. "Auto-Airfoil" includes advanced graphics so that displays of geometry, temperatures, etc., can be monitored. Empirical data information and heat transfer correlations are included in the cooling analysis module. Thus, the most advanced analytical methods for prediction of the gas-side heat transfer are not a part of the analysis at this stage in its development. However, the method's ability to optimize geometry and render sensitivity studies through rapid iterative calculations makes it an extremely valuable design tool. Formerly it took long arduous efforts by engineers to arrive at an optimal design from an array of design proposals. Now the computer does this efficiently and reliably in a matter of minutes. There are plans to include a Life Prediction Module in a future version of this design method. It is anticipated that this computational design method will undergo revision as improved predictive methods evolve from research efforts. Empirical and correlation elements in the programs will be replaced by verified analytical models.

This section on Computational Predictions of

Heat Transfer has discussed the geometrical complexity of the flow in a turbine channel and how Navier-Stokes solvers can model the three-dimensionality of steady flow. Such flow models will allow solutions to the energy equations resulting in local predictions of heat transfer. In the turbine there is a further significant factor, unsteady flow, which is considered to be a major influence on the accurate prediction of gas-side heat transfer. The next section will address this topic.

FLOW UNSTEADINESS ANALYSIS

The physical unsteady character of the flow entering the turbine inlet nozzles has been recognized for a long time. How to account for the unsteadiness and portray it analytically in fluid mechanics and heat transfer models is a very challenging task. It would be theoretically possible to write down a complete set of the Navier-Stokes equations which would include all of the time-unsteady characteristics of the flow. However, no computer existing today has the capacity and endurance to crunch out numbers from such a general set of equations with innumerable terms.

The unsteady problem is made tractable by reducing the number of terms in the equations through an averaging procedure of both length and time scales. The trick is to select the procedure that least affects the unsteady information yet allows a solution to converge through the computer. In the averaging process a large array of unknown terms are introduced which must be investigated numerically or experimentally if closures of the "averaged" equations are to be achieved. There is the further problem of verification of the entire analytical procedure by an experimental test. The capacity to simulate the unsteadiness and to measure it properly are limitations of verification. Consequently, one has to weigh the analytical averaging procedure against the available experimental technique before making a final decision on how to proceed.

A number of methods for modeling the complex flow within the turbine passages are being pursued. Reference 8 describes a study in which a finite difference, unsteady, thin-layer Navier-Stokes approach is being used to represent the interactive flow between blade rows. The relative motion between the stators and the rotor are simulated by the use of patched grids that move relative to one another. The calculational procedure produces values of time-averaged surface pressures, time-wise pressure fluctuations, and velocity information.

An interesting proposal for a method of averaging the Navier-Stokes equations has been proposed by John J. Adamczyk in reference 9, which he calls "a passage-averaging" technique. Through a three-step averaging process, random and periodic unsteadiness effects are removed from the equation set but many unknown terms emerge that have to be evaluated analytically or experimentally. Initial tests of the approach look promising. Much experimental and analytical work has to be done before a conclusive assessment of the approach is made.

The major U. S. Manufacturers of aviation gas turbine engines have embarked on research programs that address the influence of unsteadiness on design practice. The objective is to quantify the perturbation caused by unsteadiness on the performance of a blade row. Current industrial design practice is

moving into the application of three-D flow solvers for turbomachinery. The ultimate goal of this design trend is the development of a three-D viscous code that includes unsteady flow effects. Expert system programming is being examined for potential use in design procedures. This might be the future route for inclusion of unsteady effects in the design procedure. At this writing, it appears that sophisticated design procedures which include computational codes that model viscous, unsteady flow effects in three dimensions are still a long ways off. Prediction of the heat transfer associated with the unsteady flow will be a still later capability. Advances in computers, software and knowledge-based systems will determine their adoption in design practice.

EXPERIMENTAL STUDY OF UNSTEADY EFFECTS

A recent assessment of the effects of flow unsteadiness on the averaged local heat transfer in turbine stages was reported by Dring, Blair, and Joslyn of the United Technologies Research Center (10). The experimental facility was a large-scale turbomachine with highly instrumented blading. Ambient air was the working fluid and the blade surfaces were heated electrically. Although the heat transfer was opposite in direction from that of an actual engine turbine, the effects of flow unsteadiness are similar. Upstream disturbances were simulated by grids. One of the interesting observations of the testing was that the upstream disturbances (presumably from the combustor) would have major influence on the first stage stators but would have a reduced influence on the downstream blade rows. The unsteady wakes promoted by the blade rows appear to be the major downstream influence on the unsteady flow field in the turbine.

Early in the '80's an experimental program was developed at the LeRC to simulate the effect of rotating blade wakes on the stagnation region of a downstream blade. A tunnel-type apparatus was developed which consisted of a spoked rotating element upstream of a highly instrumented cylinder as the heat transfer test section. The rods in the rotating wheel produced the wakes which impacted on the heat transfer test section downstream. Both steady-state and transient methods for determining heat transfer results have been employed. In a recent reference 11, O'Brien reported results obtained on a test cylinder comprising a ceramic core and a thin outer sheath or film. The instrumented thin film was used as a heat transfer gauge in transient studies or as a part of an anemometer circuit in steady flow experiments. The latter method was considered less accurate than the former because of calibration difficulties.

Large, local instantaneous bursts in heat transfer were noted as a result of the impacts of wakes. For some conditions the level of heat transfer exceeded three times the normal inter-wake level. The wake passing effects had a pronounced influence on the time-averaged heat transfer levels as well. An important parameter in this case was the Strouhal Number. Enhanced levels in average heat transfer of approximately 30% were observed at the higher Strouhal Numbers

but diminished at the lower Strouhal values. The authors were able to correlate the steady state stagnation line data applying the method of Lowery & Vachon, reference 12.

The authors of reference 13 conducted an experimental study of the effect of forced, freestream unsteadiness on turbulent boundary layers, in an effort to characterize the boundary layer for modeling purposes. They confirmed that the time-averaged behavior of the flow was invariant with the forced frequency of steadiness and quite similar to steady flow under similar mean conditions. In contrast, the dynamic responses of the velocity and turbulence were strongly affected by the unsteadiness frequency. They concluded that considerations of the unsteady energy budgets were necessary in modeling the dynamic turbulence. In the analytical portion of the report they developed formulations of the equations of fluid motion into three distinct decompositions of velocity and pressure, namely: time-averaged, organized unsteady and turbulent. Apparently, the analytical task was carried out first before embarking on the experimental program. A two-color laser doppler system was employed as the primary diagnostic instrument.

One can conclude that there is a sizeable national research effort devoted to the effects of flow unsteadiness on turbomachinery aerodynamics and heat transfer. Most of this effort addresses the aerodynamic issues. In the heat transfer area, a prime concern is the state of the boundary layer--laminar, transition or turbulent. In an unsteady environment, there are questions on how to characterize the turbulent structure of the boundary layer because the eddy structure is based upon a data base of experimental observations in a relatively quiescent flow environment.

BOUNDARY LAYER TRANSITION

In 1984, the Lewis Research Center sponsored a conference entitled "Transition in Turbines". The proceedings of the conference is contained in reference 14. Included in the attendees were transition research. At this conference, Mark Markovin, Professor Emeritus of Illinois Institute of Technology, gave the keynote lecture. Professor Markovin is credited with proposing a mechanism of transition that occurs in place of low disturbance theory. He labeled the mechanism "bypass" because the transition process associated with classic low disturbance stability theory is bypassed. Experimentally, the "bypass" transition was noted unexpectedly in some of the early reentry tests of the 1950's. If "bypass" transition can occur in a steady flow field, it is certainly the probable mechanism of transition in an unsteady flow field such as occurs in a gas turbine flow path. Professor Markovin's lecture was an elucidation of the physical effects and parameters which are thought to be influential in bypass transition. The conference concluded with strong recommendations for the pursuit of experimental research in a "hierarchy of facilities" ranging from bench testing to large facilities that simulate the real environment of the gas turbine engine. A plea was made for a major effort in instrumentation research and development in support of the overall experimental research program.

Another important recommendation from the conference suggested a multifaceted analytical effort. The best simulation of the transition process was

thought to be the full Navier-Stokes approach followed closely by the Large Eddy Simulation (LES). Empirical models will continue to be useful but will lack generic applicability to a range of flow conditions and geometries.

In a recent publication (15) the authors reported on a study in which the effects of pressure gradient and to a limited extent the effect of turbulence intensity on transition were observed in a wind tunnel experiment. It was observed that pressure gradient, whether negative or positive, did influence transition. Within the limits of varying turbulence intensity in the experiment, very little effect of turbulence on transition was observed.

In contrast to these results Blair (16) completed a set of windtunnel tests in which turbulence levels ranging from 0.7 to 5% were imposed and he observed a significant effect of turbulence levels on transition. It is difficult to reconcile the differences in the somewhat contradictory results except to recognize that the apparatuses and test conditions were different for the two investigations. I tend to agree with the view that upstream turbulence does markedly affect the location of transition from personal experience. Blair did show that acceleration rate altered the transition which also agrees with the observations in reference 15.

Experimentation in a near-engine environment is needed to elucidate acceleration/deceleration and turbulence effects experienced in an actual engine.

Another recent effort to develop a semi-empirical analytical model of transition was reported in reference 17 by Patankar and Schmidt of the University of Minnesota. They modified the Lam-Bremhorst low Reynolds-number k-E turbulence model to account for pressure gradients. Comparing the predictions of their model with experimental measurements of heat transfer in turbine geometries, they were encouraged by the improved predictive capabilities of their analytical model. However, improvement in the predictions might result by further modification of the parameters in the turbulence model to account for dissipation in the near-wall region and curvature effects.

One of the most reliable experimental methods for detecting transition is through local measurement of heat transfer. Michael Dunn (18) has made detailed measurements of the local heat transfer on turbine blades using thin film gauges. He employs a shock tube technique to provide a short duration of high temperature gases that enter an actual turbine or test model thereof. From thousands of data samples he has developed profiles of the heat flux that can be translated into three-dimensional plots of the heat transfer. In graphical form, these plots reveal the distribution of the transition loci. The high response instrumentation has enabled detection of the periodic heat transfer fluctuations caused by the wake interaction between the rotor and stator blade rows. These data gave clear evidence that the transition region oscillated so that local areas changed from laminar to turbulent and visa versa over a span of time. Dunn's research is an important contribution to the understanding of unsteady phenomena and transition in turbines.

At the Lewis Research Center, a small closed circuit wind tunnel has been built which is being used for an investigation of transition phenomena as it occurs on turbine blades or vanes. One of the principal objectives is to uncover the difference in the triggering mechanism between low disturbance transition and larger disturbance perturbation of a laminar boundary layer. As noted earlier, the former is referred to as classic transition while the latter is called "bypass" transition. Early results discussed in reference 19 indicate that even at low turbulence levels of the freestream, it is possible to experience the bypass mode of transition. No criteria have been developed as yet which predict the onset of this type of transition, over a broad range of conditions.

Although 5 years have elapsed since the "Transition in Turbines" conference took place, most of the issues considered remain as problem areas warranting further investigation. Development of the physical understanding and the associated predictive codes remain in the agenda of needed research. One of the strong recommendations that evolved from the turbine transition conference pertained to well-defined physics so that the periodic boundary conditions for such modeling schemes as Large Eddy Simulation are realistic. The complex Large Eddy Simulation is more likely to successfully portray external flow over aerodynamic surfaces. Depicting turbulent transition in internal flows by this means appears extremely challenging. However, LES and other sophisticated analytical approaches must be tried if empiricism in the modeling is to be reduced and more generic models result.

CURVATURE EFFECTS

Curvature effects have been recognized as a contributing influence to the levels of heat transfer in turbulent flow. Concave curvature introduces strong destabilizing effects in turbulent boundary layers which induce higher levels of heat transfer than are experienced along flat plates. In contrast, convex curvature tends to stabilize a turbulent boundary layer with resulting reductions in heat transfer levels and wall friction. The presence of severe concave and convex curvatures in turbine flow passages necessitates accounting for curvature in estimating the gas-side heat transfer levels on blade surfaces. Few of the flow regions in a turbine passage can be characterized by a flat plate or straight channel. One of the major questions pertaining to turbine heat transfer pertains to the effect of curvature on boundary layer transition. An even more complex issue is how curvature and flow unsteadiness are coupled to create the stability environment that controls whether the boundary layer is laminar-like or turbulent. In reference 20 experiments were reported that included both upstream levels of turbulence and convex curvature beyond a nozzle approach section. Among the important findings were the following:

- a) Convex curvature delays transition.
- b) For the cases studied, the influence of the upstream turbulence levels was greater than the curvature effects.
- c) Convex curvature is more influential in reducing the heat transfer than the skin friction. Stated differently, the Reynolds analogy factor for convex curvature is lower than the flat wall case. The experimental results from reference 20 further

confirm that curvature greatly complicates the prediction of heat transfer from the hot gas to the turbine blade. More experimental studies and analysis are required before reliable generic heat transfer models will be available for design purposes.

HOST PROGRAM

In the Introduction, mention was made of the HOST program and its role in addressing some of the major heat transfer problems of turbine technology. Several of the research programs sponsored by HOST have been mentioned or referenced in sections of this paper. A general review of the heat transfer research accomplishments is found in reference 21. However, the general contribution of this program warrants special recognition in a separate section. At the conclusion of the program, an independent evaluation was sponsored. The evaluation addressed the entire HOST program which encompassed many topics besides turbine heat transfer. A careful survey of university, industry, and government participants revealed a unanimous acclaim that the program was very effective and to the mutual benefit of all the participants. Most would have liked the program to continue. When it was in operation, it was an effective forum for technical exchange. The annual workshops were cited as especially valuable forums. These were well attended by the most respected researchers in the principal discipline areas, and the exchange of information was well administered. Many felt that improved relations were cultivated among industry, university, and NASA representatives under the auspices of HOST.

Approximately three quarters of the funding was allocated to experimental investigations. A number of benchmark quality data sets (for example reference 22) have been produced which will continue to be valuable resources for validation of new or upgraded computational codes. It is unfortunate that the cancellation of the HOST program has stopped the valuable annual workshops where data sets were announced and shared.

CONCLUSIONS

Computational

Fluid mechanics is fundamental to the prediction of heat transfer within the turbine. Therefore the advances in CFD are a major determinant in advancing the methodology of estimating heat transfer. The CFD formulation must include the energy equation, if predictions of heat transfer are to evolve from the calculation.

Over the past decade much progress has been made in the area of computational modeling of flow through turbine passages. Replacing the early one-D or two-D computational procedures are more comprehensive two-D and three-D programs some of which involve treatment of the Navier-Stokes equations. In fact, considerable progress is being made toward the development of three-D viscous formulations. In these advancements, the inclusion of unsteady flow effects is also in progress.

Another significant advance relates to

efforts to comprehend blade-to-blade interaction effects in a computational scheme. Considerable progress has been made with computational programs of this type on class 6 computing facilities. Particularly impressive have been the graphical results from the operation of this type of program.

The prediction of transition in turbomachinery has been recognized as an important computational problem. More extensive evaluations of empirical or semi-empirical correlations are being checked out against the available data bases. Large Eddy Simulation is a more analytical approach that is being pursued to represent the transition phenomenon. Some research specialists in transition advocate modeling of transition by utilization of the full-blown Navier-Stokes equations and are attacking the problem in this manner. While transition appears to be receiving more attention of late, there doesn't seem to be anything specific that can be singled out as a major milestone of achievement.

Geometric effects, such as curvature, are being accommodated in computational schemes through empirical or semi-empirical corrections to turbulent transport diffusivity terms. These corrections are useful but no generic methods are available.

The general effort in CFD is out in front of the design application, as it should be. All that is accomplished in CFD can't be utilized in design procedures as a practical matter. However, the advances in computational methods serve the important function of leading the technology and introducing the physics into the design schemes. There is a great need to have those who can translate the advances in the computational area into workable design procedures.

CONCLUSIONS

Experiments

Several outstanding data sets from cascades, tunnels and large scale turbo-machines are available for use by researchers in the disciplines of heat transfer and fluid mechanics. These data sets are characterized by substantial improvements in the quality and detail over prior results. This improvement can be attributed to the use of sophisticated, non-intrusive instrumentation, more rapid data acquisition and on-line monitoring of the data through direct connection to a computer. Another important factor was the development of special facilities that were dedicated to a particular type of experiment. In many cases, the facility was expressly built for the experiment to simulate an important phenomenon under investigation.

The recent advances in CFD modeling rather than displacing the need for continued experimentation, has established newer and greater demands for more experiments. It has become necessary to insist on the development of sets of "bench mark" experiments specifically designed to match the boundary conditions and initial conditions of CFD models so that these models can be verified. Such "bench mark" experiments not only check out the physics of the problem, they may uncover phenomenon not anticipated in the formulation of the model. Verification will continue to be a major activity as more complex CFD models are developed. In addition, before these models are accepted as elements of a design scheme, checkout of the predictions at near-operating conditions will be demanded. As mentioned in reference 12, there is a

hierarchy of experimental facilities that enable verifications to be done from the introduction of an idea until it proves to be a serious candidate in a design procedure. Moving along this course of facilities, the experimental conditions move closer to the thermodynamic state of the actual engine operation. Simplicity and ease of performing the experiment are greatly reduced as the experimental conditions approach engine conditions.

FUTURE CHALLENGES & RECOMMENDATIONS

The continuing pursuit of programs that describe the complicated three-D flow in a turbine is a priority need. A primary goal should be the achievement of three-D viscous flow, Navier-Stokes computations suitable for application to a turbine passage which include unsteady flow effects. A computer program should result which can be operated at a work station available to an engineer at his/her work station. Such a program must have the capability of computing viscous losses and heat transfer. Before such a program is accepted it must undergo extensive scrutiny in rigorous verification experiments in addition to the debugging exercises of the computer program itself.

The analytical schemes that are directed at predicting the effects of blade row interactions are very innovative and look promising but much more evaluation is required. Again verification in suitable experiments is a priority of this area of research.

In modeling the unsteady flow component of the three-D Navier-Stokes equations, the upstream effects from the compressor and combustor that introduce disturbances in the flow, must be accounted for. These are unique disturbances that are difficult to represent in the more conventional way as a form of turbulence.

At present there aren't reliable data sets that represent them adequately. It will be a real challenge to develop them.

There is a real need to develop more generic analytical models for predicting the fluid flow and transport characteristics of flow situations where curvature or acceleration/deceleration are encountered.

The prediction of the onset of transition on the aerodynamic surfaces of blade rows is another challenging area that is still unresolved. The influence of upstream disturbances, blade row interactions etc. greatly complicate this prediction requirement. There is also a major experimental challenge in transition research. Data is needed from operating conditions that approximate the engine operating conditions; this will be extremely difficult to acquire at elevated temperatures and pressures.

Even with the advent of new high temperature materials, film cooling of segments of the blades will continue to be necessary. The complex interaction of the film coolant stream with the hot gas stream is difficult to model. Certainly the magnitude of the coolant must be minimized so as not to penalize the performance of the engine. Therefore, it is important to continue to pursue reliable methods of predicting surface thermal

conditions in the regions of film cooling. True, sizeable advances have been made over the past decade, but there is still a ways to go.

All of the advances in analysis and experimentation will be for naught unless they can be used to modify or influence the design procedure for advanced engines. An all important link in the chain is the person who can translate the research advances into workable design procedures and can make use of the available data bases to verify their legitimate application in design.

The HOST program provided the opportunity for the turbine research community along with others involved in engine hot section problems to meet together and have meaningful discussions and share data, computer programs and viewpoints, in general. The cancellation of HOST has eliminated such planned interactions. Some means ought to be found to restore these meeting opportunities so that these interactions can be resumed.

Looking over the past decade, considerable progress has been made in addressing the fundamental heat transfer problems mentioned in reference 1 that apply to accurate methods of predicting the thermal loads in aircraft gas turbines. Progress can be identified in both experimental and analytical research. However, it is evident that there is much more to do to bring about a higher level of confidence in predictive design methods for the advanced engines of the future. With pride in what has been accomplished, the aero-thermo research community can continue its mission of acquisition of new knowledge through aggressive research. Major challenging opportunities lie ahead for those who wish to participate.

REFERENCES

- 1) Graham, R. W. (1979)
"Fundamental Mechanisms That Influence the Estimate of Heat Transfer to Gas Turbine Blades," NASA TM 79128.
- 2) Crawford, M. and Kays, W. (1976)
"STAN 5 - A Program for Numerical Computation of Two-Dimensional Internal and External Boundary Layer Flows," NASA CR 2742.
- 3) Yang, R. J.; Weinberg, B. C.; Shamroth, S. J and McDonald, H. (1985)
"Turbine Vane External Heat Transfer. Vol. 2 - Numerical Solutions of the Navier-Stokes Equations for Two- and Three-Dimensional Turbine Cascades with Heat Transfer," NASA CR 174828.
- 4) Chima, R. V. and Yokota, J. W. (1988)
"Numerical Analysis of Three-Dimensional Viscous Internal Flows," NASA TM 100878.
- 5) Knight, C.J. and Choi, D. (1987)
"Development of a Viscous Cascade Code Based on Scalar Implicit Factorization," AIAA Journal, Vol. 27, No. 5, pp. 581-594.
- 6) Choi, D. and Knight, C.J. (1988)
"Computation of 3-D Viscous Cascade Flows," AIAA Paper 88-0363.
- 7) Andrews, G. V. and Starkweather, J. H. (1988)
"Heat Transfer Design with Auto-Airfoil," The Leading Edge, Aerodynamic Research at GEAE (General Electric Bulletin), Winter, 1987-1988.
- 8) Rai, M. M. (1987)
"Unsteady Three-Dimensional Navier-Stokes Simulations of Turbine Rotor-Stator Interactions," AIAA Paper 87-2058.
- 9) Adamczyk, J. J. and Graham, R. W. (1986)
"Numerical Simulation of Multiblade Row Turbomachinery Flows," Cray Channels, Vol. 8, Summer, pp. 12-17.
- 10) Dring, R. P., Blair, M. F. and Joslyn, H. D. (1986)
"The Effects of Inlet Turbulence and Rotor-Stator Interactions on the Aerodynamics and Heat Transfer of a Large-Scale Rotating Turbine Model. Vol. 3, Heat Transfer and Data Tabulation 65% Axial Spacing," NASA CR 179468.
- 11) O'Brien, J. E. (1988)
"Effects of Wake Passing on Stagnation Region Heat Transfer," Heat Transfer in Gas Turbine Engines and Three Dimensional Flows, E. Elovic, J. E. O'Brien, and D. W. Pepper, Eds., ASME, pp. 17-28.
- 12) Lowery, G. W. and Vachon, R. L. (1975)
"The Effect of Turbulence on Heat Transfer from Heated Cylinders," International Journal of Heat and Mass Transfer, Vol. 18, No. 11, pp. 1229-1242.
- 13) Brereton, G. J. and Reynolds, W. C. (1987)
"Experimental Study of the Fluid Mechanics of Unsteady Turbulent Boundary Layers," Report TF-29, Thermosciences Division of Mechanical Engineering, Stanford University. (Avail. NTIS, AD-A184856).
- 14) NASA (1984)
Transition in Turbines, NASA CP 2386, National Aeronautics and Space Administration, Washington, D.C.
- 15) Fraser, C. J., Milne, J. S., and Gardner, I. D. (1988)
"The Effect of Pressure Gradient and Freestream Turbulence Intensity on the Length of Transitional Boundary Layers," Institution of Mechanical Engineers Proceedings, Part C, Mechanical Engineering Science, Vol. 202, No. C3, pp. 195-203.
- 16) Blair, M.F. (1982)
"Influence of Free-Stream Turbulence on Boundary Layer Transition in Favorable Pressure Gradients," Journal of Engineering for Power, Vol. 104, No. 4, pp 743-750.
- 17) Schmidt, R. C. and Patankar, S. V. (1987)
"Prediction of Transition on a Flat Plate Under the Influence of Freestream Turbulence Using Low Reynolds-Number Two Equation Turbulence Models," ASME Paper 87-HT-32.
- 18) Dunn, M. G., Martin, H. L. and Stanek, M. J., (1986)
"Heat Flux and Pressure Measurements and Comparison with Prediction for a Low-Aspect-Ratio Turbine Stage," Journal of Turbomachinery, Vol. 108, No. 1, pp. 108-115.
- 19) Suder, K. L., O'Brien, J. E. and Reshotko, E. (1988)
"Experimental Study of Bypass Transition in a Boundary Layer," NASA TM 100913.
- 20) Wang, T. and Simon, T. W. (1987)
"Heat Transfer and Fluid Mechanics Measurements in Transitional Boundary Layers on Convex-Curved Surfaces," Journal of Turbomachinery, Vol. 109, No. 3, pp. 443-452.
- 21) Gladden, H. J. and Simoneau, R. J. (1988)
"HOST Turbine Heat Transfer Program Summary," NASA TM 100280.
- 22) Nealy, D. A., Mihelc, M. S., Hylton, D., and Gladden, H. J. (1984)
"Measurements of Heat Transfer Distribution Over the Surfaces of Highly Loaded Turbine Nozzle Guide Vanes," Journal of Engineering for Gas Turbines and Power, Vol. 106, No. 1, pp. 149-158.



National Aeronautics and
Space Administration

Report Documentation Page

1. Report No. NASA TM-102460		2. Government Accession No.		3. Recipient's Catalog No.	
4. Title and Subtitle Recent Progress in Research Pertaining to Estimates of Gas-Side Heat Transfer in an Aircraft Gas Turbine				5. Report Date	
				6. Performing Organization Code	
7. Author(s) Robert W. Graham				8. Performing Organization Report No. E-5247	
				10. Work Unit No. None	
9. Performing Organization Name and Address National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191				11. Contract or Grant No.	
				13. Type of Report and Period Covered Technical Memorandum	
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, D.C. 20546-0001				14. Sponsoring Agency Code	
15. Supplementary Notes Prepared for the 1990 Turbo Expo sponsored by the American Society of Mechanical Engineers, Brussels, Belgium, June 11-14, 1990.					
16. Abstract A decade ago several important fundamental heat transfer phenomena were identified which were considered basic to the ability to predict heat transfer loads in aircraft gas turbines. The progress in addressing these fundamentals over the past ten years is assessed in this paper. Much research effort has been devoted to their study in university, industry and government laboratories and significant progress has been achieved. Advances in computer technology have enabled the modeling of complex three-D fluid flow in gas turbines so necessary for heat transfer calculations. Advances in instrumentation plus improved data acquisition have brought about more reliable data sets. While much has advanced in the '80's, much challenging research remains to be done. Several of these areas are suggested in the paper.					
17. Key Words (Suggested by Author(s)) Gas turbine heat transfer			18. Distribution Statement Unclassified—Unlimited Subject Category 07		
19. Security Classif. (of this report) Unclassified		20. Security Classif. (of this page) Unclassified		21. No. of pages 8	22. Price* A02