General Disclaimer

One or more of the Following Statements may affect this Document

- This document has been reproduced from the best copy furnished by the organizational source. It is being released in the interest of making available as much information as possible.

- This document may contain data, which exceeds the sheet parameters. It was furnished in this condition by the organizational source and is the best copy available.

- This document may contain tone-on-tone or color graphs, charts and/or pictures, which have been reproduced in black and white.

- This document is paginated as submitted by the original source.

- Portions of this document are not fully legible due to the historical nature of some of the material. However, it is the best reproduction available from the original submission.

NASA Technical Memorandum 83348

Opportunities for Research in Aerothermodynamics

Robert W. Graham
Lewis Research Center
Cleveland, Ohio

NASA

OPPORTUNITIES FOR RESEARCH IN AEROTHERMODYNAMICS

Robert W. Graham

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

INTRODUCTION

The title of this talk includes the term "aerothermodynamics" which may be unfamiliar to some of you. I am using the term "aerothermodynamics" to signify a combination of basic disciplines including fluid mechanics, aerodynamics, chemical kinetics, heat transfer and thermodynamics. All of these pertain to the gas turbine engine. My purpose in this lecture is to present arguments in support of more active fundamental research programs involving these disciplines as they apply to gas turbine engines. Such research is justified as a means of supplanting current empirical design procedures with newer and more reliable design methods that incorporate validated computational approaches. By elevating the process of design to a higher, scientific level, expensive, long-term trial and error processes with actual hardware will be minimized. From documented aircraft turbine engine development experience, it has been observed that anywhere from 10 to 40 percent of the expense of engine development is tied up in the core turbine alone, figure 1. Depending on the engine, 30 to 300 million dollars can be expended on cooling design changes to the turbine. In addition, valuable time is lost in cut-and-try revisions of the components which adds to the burden of cost.

I don't mean to infer that there is no engineering base to the current engine design processes. However, there is ample opportunity to enhance and upgrade the design process through the introduction of research-derived information. Stronger collaboration among industry, universities and government could result in such a significant achievement.

Those of us who have been active in this general area of "aerothermodynamics" recognize that many of the questions or research topics have been identified for a long time. Twenty or thirty years ago, if I appeared here, most of these issues I will mention today would have been mentioned then. However, several major changes have occurred in the recent decades.

As I have already mentioned, the cost of developing new engine hardware has escalated so sharply that the industry can no longer afford to be satisfied with cut-and-try procedures. There is a strong economic incentive for greater design assessment before parts are fabricated.

The design procedure for the future can be made more sophisticated because of two major developments:
(a) The computer and computational methods are powerful means of modeling physical phenomena. The engineer can transcend the restriction of "correlation only." The speed of the computer facilitates iterative solutions so that parametric design changes can be observed "on paper" before the final design judgment is made.
(b) Major developments in instrumentation techniques and rapid, high-capacity systems for recording and processing experimental data.

In the process of achieving an improved design procedure, it must be recognized that there is an orderly evolvement of information and knowledge. If the design process can be thought of as a structure, there are

building blocks which are necessary for a sound structure, figure 2. The foundation has to be scientific understanding -- which is the base of all engineering knowledge. The next level above the base is "physical modeling." In this step, clever experiments and analyses must be devised which represent phenomena assumed to be important. Achieved in this step is an evaluation and prioritization of phenomena that warrant further investigation. The third level builds on the most promising experiments and analyses of the lower level. The necessary ties between the experiments and the analyses are strengthened. Models evolve from a rigorous verification process of comparison between analyses and selected experiments.

Before a model can become part of a design procedure, it must undergo further scrutiny and testing. The third level involves experiments that simulate important effects expected in the real world of the engine. However, the simulation may be done at conditions that don't completely represent the real world in scale or thermal conditions, for example. Consequently, the next step is to further verify the model at more severe conditions: if at all possible at conditions that closely represent the real engine.

The final level or cap to this structure is the incorporation of the model into a design code that is utilized in advanced engine design. There has to be "feedback" among the four elements of this structure before the design code is established.

Within the time constraints of this lecture, I am going to review some of the principal research topics that could lead to greater certainty in design. To a considerable degree, I will depend on my familiarity with NASA-sponsored research programs in these areas; but other sources of university or industrial research will be included. This is not intended to be a comprehensive survey article. The status of some current research areas will be reported to indicate opportunities for further research in future programs.

COMBUSTION

As is vividly portrayed in a schematic, figure 3, the combustor is a highly complicated flow and reaction system. Actually it can be considered to be made up of several subsystems involving: (a) fuel injection, (b) air/fuel reaction and (c) dilution of the combustion products. In fundamentally-oriented research programs, these three are generally studied separately.

Fuel Injection

For the fuel injection process, the information of interest is the statistical size, density distribution and the velocity of the droplets in the injection zone. In recent years, laser optical systems have replaced laborious droplet tracking and identification techniques. Systems are commerically available that make it possible to determine statistical droplet size and estimate spacial density distribution by analyzing the output of a laser beam passing through a spray pattern. One approach makes use of the Rosin-Rammler equation which was developed for predicting distributions of powders. The results must be correlated with the injector geometry, pressure drop, property of the liquid fuel and the consideration of intervening mechanisms such as vaporization. It is highly desirable to put in place improved, predictive analytical schemes that characterize the liquid

2

sprays as they enter the combustion zone. New experimental methods such as the Dual Beam Light Scattering Interferometer shown in figure 4 hold promise of providing more meaningful experimental data to support the analytical effort.

Computational Modeling of Combustion

There is no more obvious example of how computational fluid mechanics has impacted aerothermodynamics than in the modeling of combustion. Many highly competent analysts throughout the United States and the world are involved in the modeling process. At the University of California, Berkeley, Professors Ghoniem, Chorin and Oppenheim have been participating in the development of a two-dimensional computer code for turbulent combustion. This is a joint effort with the Lewis Center and John Marek of the Combustion Branch.

In principle, the method involves the coupling of the conservation equations for fluid mechanics with equations for chemical kinetics. Such a system of equations is non-linear and elliptic in form. The statistical nature of the initial and boundary conditions further complicate the analysis.

Despite these difficulties, this joint effort has been successful in developing a computational method. In essence, the computation is carried out in two major parts: (a) vortex dynamics and (b) combustion kinetics. A random vortex method originally developed by Chorin allows a turbulent field to develop as vortex elements in the overall flow. In the combustion part, the flame front is advected by the velocity field. The flame front is further displaced by the corresponding burning speed of a laminar flame.

The computational method can be applied to a number of two-dimensional geometries. Of particular interest is the backward facing step. Comparisons of visual observations of real combustion in such a geometry have been made to computational graphics of the combustion process. The similarities in the comparative structure of the turbulence are encouraging. Figure 5 is a computed progression of the flame front. Extending such an approach to a three-dimensional flow field poses an opportunity and a challenge. As in most of the research topics discussed, I have presented one approach. Undoubtedly, other approaches are being considered or will be developed. The step increase in the capability afforded by Class VI computers is an enticement which won't be overlooked in combustion modeling. It poses a great opportunity to those skilled in computational fluid mechanics.

Dilution Zone Mixing

The proper thermal conditioning of the products of combustion is accomplished by dilution with air. The dilution process influences the average temperature level and uniformity of that level as the gases leave the combustor. It is desirable for the combustor designer to have some analytical tools that enable him to prescribe the location and flow rate of the individual dilution jets.

In its simplest geometry, the dilution process can be portrayed as jets penetrating normally into a duct flow. Aerojet Corporation carried out an extensive experimental study of this configuration and a correlation for the dilution process resulted. Recently, an interactive numerical code was developed based on the correlation. It is capable of predicting the dilution temperature field at any station downstream of the jet penetration, figure 6. Computer graphics enable the temperature profiles to be plotted

3

in three dimensions. While such an approach is useful, it can't be applied to situations in which the geometry or conditions differ markedly from the experiments which are the source of the correlation.

Analytical work is also progressing in the development of a three-dimensional computational model of dilution which is based on the Navier-Stokes equations. The two-equation model of turbulence is applied. Figure 7 shows velocity and temperature profiles developed at two planar stations downstream of the jet. Approximately 70,000 grid points were used and over six hours of IBM 370 time were required for each plot. Class VI and VII computers will reduce this time by approximately an order of magnitude.

This is a specific example of complex, three-dimensional flow caluculations that the growing capability of computers will enable engineers to carry out in a routine fashion.

Combustor Heat Transfer

The radiation component of heat transfer is a major consideration in combustor liner design. Conventional design practice makes use of air cooling the combustor liner by convection and film cooling, see figure 3. Under consideration are ceramic coatings and transpiration materials for the liners. Actual heat transfer measurements, as shown in figure 8, have demonstrated that the hydrogen/carbon ratio of the fuel is an important influence on the level of radiation flux. Any reduction in the hydrogen/carbon ratio of the fuel results in significant increases in radiation flux. This observation is particularly important because there is a trend in fuel refining toward lower hydrogen content because of the condition of the crude stocks. Also, the synthetic hydrocarbon fuels of the future made from shale or coal will definitely be lower in hydrogen/carbon ratio.

From this observation, it appears that there is an opportunity to engage in research which includes a combination of radiation and convective heat transfer in combustion systems.

Before leaving the topic of combustion, it is important to remark that more experimental information about the condition of the gases discharging from the combustor is a major research need. As will be pointed out later, actual velocity/temperature profiles and turbulence structure are essential ingredients in the estimates of turbine heat transfer. Very little quantitative data on real combustor conditions are available.

TURBINE AERODYNAMICS

The turbine appears to be a very simplistic machine. On closer examination, it poses many tough engineering problems. Among these listed in figure 9, there are a number of very important problems which are a part of "fluid mechanics." In general, they can be classified as the ability to describe the complex 3D flow field and predict the viscous losses associated with these flows.

Up until recently, the computational methods available for turbine applications were limited to pseudo-3D inviscid codes for subsonic flow. Known as MERIDL and TSONIC, these codes enabled computation on the hub-to-tip midchannel flow surface and the blade-to-blade stream surfaces, respectively (see fig. 10). These have been widely adopted by industry and adapted to design needs.

Coming into place today are inviscid 3D codes for turbine flow channels which can be run in reasonable computer times. Efforts are underway to

verify their predictions by laser anemometry in 3D annular turbine cascades as depicted in figure 11. Note that Denton's 3D inviscid code agrees closely with the measurements. This 3D code is also being exercised in predicting what happens to the combustor exit radial temperature profile as the gases procede through the turbine. Early results show a moderate rotation of the profile in the rotor. Experimental verification of the profile variation is planned at the Lewis Center.

The researchers who have been rendering this comparison in the cascades have succeeded in measuring the third velocity component along the optical axis. Generally with the laser anemometer, only the velocity components normal to the optical axis are measurable. They incorporated the principles of the Fabry-Perot interferometer to enable the optical axis component measurement.

One of the principal difficulties in developing 3D codes for turbines has been the grid selection which must be patterned for the high turning and convergence of the channels. It should be recognized that although the large turbines for aircraft applications are generally axial, smaller turbines are frequently radial or mixed-flow machines. There is much interest in these smaller machines for both civilian and military applications. Their inherent complex geometry and the inclusion of curved ducting and scrolls in the installation make the small turbine difficult to analyze. An example of a turbine ducting mesh is shown in figure 12. Professor Hamed of the University of Cincinnati has been developing a 3D inviscid finite element code for turbine scroll inlets. However, the scaling effects may make viscous effects more dominant in small turbines. Consequently, it appears that 3D viscous codes may be a requirement for the design of small turbines. Efforts are underway to develop these codes for turbine application. There are good opportunities today for contributions to analytical or computational methods suited to the turbine geometry.

As a final comment to this section; whenever experiments are designed to verify the computational codes, extreme care must be exercised to replicate the initial conditions and boundary conditions of the analysis. It is highly advisable for the analyst and the experimentalist to agree ahead of time on the initial and boundary conditions so that a meaningful verification process will be pursued.

TURBINE HEAT TRANSFER

Gas-Side Heat Transfer

The prediction of the gas side heat transfer on the exterior of turbine blades is fraught with several major uncertainities, figure 13. As mentioned in the Combustion section, characterization of the gases entering the turbine is a major initial condition requirement for turbine heat transfer estimates.

The stagnation region of the vanes and blades of a turbine are critical surfaces to be cooled. As in any cross flow situation which involves a cylinder-like body, a boundary layer is initiated near the stagnation locus and continues its development in the streamwise direction. Although much research has been devoted to investigating the fluid mechanics and heat transfer of crossflow over a cylinder, several fundamental questions relevant to turbine heat transfer remain unanswered. One of the principal questions is the influence of the upstream turbulence on the structure and growth of the developing boundary layer. A recent study by Professor Robert

Mayle of RPI has documented the effect of a small perturbation in velocity on the local heat transfer in the stagnation region. His experimental approach involves the mass transfer of napthalene as a measure of heat transfer. As indicated on figure 14, a one-percent perturbation in free stream velocity caused by an upstream screen grid produced a 10% change in local heat transfer (the Sherwood number).

If relatively small flow disturbances can effect amplified changes in stagnation heat transfer, then large upstream disturbances found in the turbine environment could promote major perturbations in stagnation heat transfer. This question is being pursued at the Lewis Center in an experimental study in which upstream disturbances are being imposed and their effect is registered on a carefully instrumented cylinder downstream, figure 15.

It is presumed that the boundary layer develops in the customary fashion at the stagnation zone first as a laminar layer and then experiences a transition to turbulent early in its development. The coupling between the free stream turbulence and boundary layer turbulence remains an unsolved question. Furthermore, the mechanism of transition within the boundary layer for a turbine environment of large disturbances is not understood. At the Lewis Center, a special tunnel is being constructed to simulate turbine-like upstream disturbances. The research data from this facility should enable the eventual introduction of more adequate transition criteria into the STAN 5-boundary layer code. Examples of the limited success with some of the existing transition criterea is shown in figure 16. Assisting in this endeavor is the well-recognized talent of Professor Eli Reshotko of Case-WRU. The transition problem in turbomachinery is completely different from that of external aerodynamics -- the topic most frequently addressed in the transition literature. The primary difference lies in the magnitude and nature of the disturbance that causes the transition. Models of transition for quiescent aerodynamic conditions involve linear representation of the growth of small disturbances. In the turbine environment, the initial disturbances are large and their growth appear highly nonlinear. Conceptually, the linear growth path is bypassed as the large disturbances cause transition. A model for this process is yet to be developed.

The flow environment of the turbine is further complicated by the presence of wakes caused by struts, guide vanes and blade rows. Such wakes contribute periodicity or unsteadiness to the flow. Efforts are underway to assess the magnitude of the wake effect in a controlled experiment. A low speed, rotating wake simulator, shown in figure 17, has been built at Lewis for this purpose. Removable cylinders have been arranged in a spoke-like configuration on a hub. Downstream, a stationary set of cylinders in radial mountings experience the wakes. Two or three of these cylinders are instrumented for local heat transfer measurements. By comparing wakeless flow conditions with the case where wakes are present, the influence of wakes on the time-averaged heat transfer will be observable.

Coolant-Side Heat Transfer

As is evident from an examination of the cross section of any typical cooled blade or vane shown in figure 18, the path taken by the coolant is highly intricate. Even the portions of the blade that are cooled convectively involve complex geometric effects. In addition, there are regions that depend on other types of cooling such as film and impingement. Consequently, the varied methods of cooling add to the geometric intricacies to yield a highly complex system of internal cooling. It has been customary

in design practice to simplify greatly the model of this internal cooling. Generally, correlations are applied in a piece-wise fashion to predict the cooling effectiveness in appropriate segments of the blade or vane. Undoubtedly, this empirical approach contributes significantly to the uncertainty in predicting wall temperature. Also, the correlations employed were probably developed in idealized test conditions of flow development, uniform wall flux or temperature and simple passage geometry. Any one or all of these restrictions may be violated to some degree in the actual condition so the estimation is a crude approximation on the coolant side of the blade.

Impingement cooling. - It is apparent that there needs to be an effort directed at developing more sophisticated means for calculating the coolant-side heat transfer. Experiments have been devised to simulate the entrance effects and hairpin bends of the cooling passages. Over the past five years, there has been a major effort at Arizona State University to study impingement cooling. Cross flow and geometric patterns of the impingement jets have been investigated in flat plate configuration as shown in figure 19. Even in the simplest geometries that can be devised, the phenomena are complex. The experimental results have been reported parametrically in correlations. It does seem possible that computational methods could be applied to formulate predictions in a somewhat analagous fashion to the dilution jet mixing anlysis mentioned in the combustion section of this presentation. If you recall the description of the interactive numerical code for dilution jets penetrating normally into a duct flow, Holdeman et al have extended a correlation into a multi-dimensional numerical model. Such an approach may be feasible in impingement cooling also. In addition to opportunities for analytical modeling, considerable experimental investigation remains open. For example, the effects of curvature and real temperature ratios warrant experimental study. More complex situations in which impingement and film cooling are coupled offer the experimentalist a great challenge.

Film cooling. - A discussion of film cooling alone could occupy the entire time allocated to this talk. There is extensive literature on the subject of film cooling. Since the 50's, studies have been made with slots, discrete holes and multiple holes for the admission of the coolant. Most of the work appears in the form of correlated data. It is often difficult to perform comparisons among the various data sources because of subtle differences in the test apparatuses which influence the free stream turbulence and the flow conditions of the coolant. Some on-going work at Lewis has shown that film cooling effectiveness is sensitive to the geometry and flow history of the coolant before it mixes with the free stream.

At this conference, we will hear a report about a research program at Stanford which examines the effect of wall curvature on film cooling. This report will illustrate a numerical approach in dealing with the experimental results from a carefully constructed experiment. Before the film cooling phase of the experiment was initiated, a thorough investigation of the boundary layer behavior of the free stream was completed. On a convex surface, the thermal boundary layer experiences a significant Stanton number reduction during the curvature. Then there is an asymptotic recovery level that is observed on a flat surface downstream of the curvature. Initial conditions upstream of the curvature have negligible effect on the asymptotic level downstream of the curvature. The curvature appears to washout any of the upstream differences.

7

As a result of the curvature effect studies, the flat plate turbulent boundary layer code known as STAN 5 has been modified to include a curvature model. This modified prediction code has been compared to a well-recognized set of experimental heat transfer data for an airfoil in figure 20. It should be noted that the predictive code does a satisfactory job of estimating the heat transfer over the convex surface of a blade.

In the process of publication at the Lewis Center is an analysis of the film cooling mixing process which includes the downstream edge of the coolant hole, the separated region and reattachment zone. Mixing length distributions were inferred from hot wire measurements of a flat plate film cooling experiment.

In film cooling research, there is ample opportunity for the development of more powerful analytical descriptions of the phenomena which takes into account multi-dimensional effects of real geometries. Film cooling promises to be one of the more important cooling schemes in future applications.

Rotational Effects

The prediction of the internal cooling heat transfer rates in a turbine rotor requires comprehension of the effects of rotation. Typical rotational speeds of aircraft turbines are of the order of 17,000 rpm and the centrifugal forces are 75,000 g's. I don't know of any heat transfer experiments that have simulated those levels. Some analyses have been performed on channels rotating about an axis perpendicular to the length dimension of the channel.

Some differences in heat transfer were noted between the leading and trailing walls of the duct, which was confirmed experimentally. The direction of the coolant flow radially outward or radially inward is also influential. Some work by W.D. Morris of Great Britain has shown that centrifugal effects enhance the convective heat transfer for radially inward flow while the opposite effect occurs in radially outward flow. His results for the radially inward case are shown in figure 21.

The effects of rotation will have to be included in the computation of the gaseous flow external to the blades in the turbine rotor. At such high "g" levels, the secondary flows will be influenced by the centripetal and coriolis forces.

CONCLUDING REMARKS

I have highlighted several research areas in aerothermodynamics that are important and challenging. There are great opportunities for clever research approaches in both analytical and experimental endeavors. In general, what is desired is a sophisticated, but user-friendly analytical approach which has been verified experimentally.

In summary, the specific problems or research areas are as follows:

In Combustion they are –
1. Characterization of the properties of sprays entering the combustion zone.
2. Computational modeling of combustion turbulence and experimental verification of the structure.
3. Three-dimensional computational descriptions of the dilution process and the resulting temperature profiles.
4. A comprehensive heat transfer and fluid mechanics model of the combustor liner.

For the above items, carefully designed experiments need to be executed to verify or qualify the analytical approaches that result from the programs. In the Turbine research areas, they are -

1. Three-dimensional viscid and inviscid computational codes for the high-turning blade rows and complex flow channels in turbine systems. Such codes must be verified in carefully executed experiments. Coordination between the experimentalist and analyst is an absolute necessity.

2. Characterization of the turbulence structure of the combustion gases entering the turbine stages.

3. An improved model of the heat transfer mechanism in the stagnation region that includes the effects of upstream disturbances, blade wakes and unsteady flow.

4. A model of boundary layer laminar turbulent transition that pertains to the highly disturbed flow environment of the turbine.

5. The practice of piece-meal application of cooling correlations in the cooling-side design of turbine blades should be upgraded into a more interactive computational method. There is need of further experimentation which simulates the entrance-like conditions and sharp turning of the cooling passages and examines the interaction of impingement, film and convective cooling schemes.

6. More comprehensive descriptions of film cooling processes which consider multi-dimensional, geometric effects that pose challenges to the analyst. As mentioned in the previous paragraph, the interaction of several cooling schemes complicates the analytical model of any one of them. In particular, expended impingement cooling air is often used in film cooling, so the models for these cooling schemes must be intimately connected in such a case. Other complicating geometric effects include wall curvature and the spacing distribution of the cooling holes. These requirements can be summed up as a three-dimensional portrayal of the film cooling process for multiple cooling holes.

Returning to a viewgraph used at the beginning of this talk, I suggested that there is a structure of engineering information that ranges from basic science to commercial design. The flow of knowledge from the basic to the commercially-applied stage doesn't always occur naturally. "Free convection" can't be relied upon to induce the transfer of information to commercial practice.

Those of us in the engineering profession are witnessing a dramatic revolution in the way engineering is practiced. The practice is transitioning from information based upon handbook guides, simplistic correlations, extrapolations and intuition to elaborate, extensive sets of information that are stored in computer files and are readily available in tabular or graphical form. In graphics, the computer-aided design process is revolutionizing engineering drafting.

As one who was trained in the old engineering school, I have feelings of being left out when I see the capabilities that recent graduates have at their finger tips in terms of their familiarity with computer software and hardware. This new capability makes a sound background in basic sciences even more essential for engineers.

The proper encouragement and support of this upward flow of engineering information to design practice is a charge that each of us here must assume. Those of you representing the university community have a unique role because you are preparing the engineer of the future to operate in this

new environment. Certainly, he or she who becomes the engineering manager of the future in industry will insist on the use of the vast resources of engineering information available from computer systems. Those of you who currently hold managerial responsibility have the opportunity and obligation to influence corporate policy toward longer-range planning and the inclusion of research as an essential overhead item.

I see those of us in government research developing centers of excellence in the basic scientific and engineering disciplines which will serve as national resources. Perhaps, the most important commitment we must assume is to cultivate the communication links and mutual support among the national laboratories, the universities and industry.

As a key element in the communication link, the government laboratories should provide the large facilities necessary to validate codes that are destined for commercial use. While we advocate the development of computational methods for fluid flow, heat transfer and combustion, we must insist that such codes be exercised against experimental conditions that simulate as closely as possible the operational conditions of future engine designs.

Even the most sophisticated codes contain empiricisms and assumptions regarding basic physical processes. Consequently, experimental verification is essential to check the physics of the analysis.

Under final checkout at Lewis is a turbine and combustor facility that is designed for verification research, figure 22. The turbine and combustor legs are independent and incorporate special instrumentation for detailed measurement. Professor Moffat, Chairman of the Thermosciences Division of Stanford University, is serving on an industry-university advisory committee to recommend the future research use of this facility. The operation of this committee is a good example of cooperation among universities, industry and government. If such a triumvirate can function effectively, the even flow of engineering knowledge from basic science to commercial practice would be assured. Certainly, in one of the channels of mobility identified as "aerothermodynamics," there are great opportunities for advancement.

<div align="center">LIST OF REFERENCES</div>

A. COMBUSTION

1. Ingebo, R.: Hydrodynamic and Aerodynamics Breakup of Liquid Sheets. NASA TM-82800, 1982.
2. Bachalo, W. D.: Method for Measuring the Size and Velocity of Spheres by Dual-Beam Light-Scatter Interferometery. Appl. Opt., vol. 19, no. 3, Feb. 1980, pp. 363-370.
3. Ghoniem, A. F., Chorin, A. J., and Oppenheim, A. K.: Numerical Modeling of Turbulent Combustion in Premixed Gases. 18th Symposium (International) on Combustion, Combustion Institute, 1981, pp. 1375-1383.
4. Holdeman, J. D. "Perspectives on the Mixing of a Row of Jets with a Confined Crossflow," submitted for presentation at the AIAA/SAE/ASME 19th Joint Propulsion Conference, June 1983.
5. Claus, R. W.: Calculation of a Single Jet in Crossflow and Comparison with Experiment. AIAA Paper No. 83-0238, Jan. 1983.
6. Claus, R. W.: Spectral Flame Radiance From a Tubular-Can Combustor. NASA TP-1722, Feb. 1981.

B. TURBINE AERODYNAMICS

1. Katsanis, T.; and McNally, W. D.: Revised Fortran Program for Calculating Velocities and Streamlines on the Hub-Shroud Midchannel Stream Surface of an Axial, Radial or Mixed-Flow Turbomachine or Annular Duct. I-Users Manual. NASA TN D-8430, Mar. 1977.
2. Katsanis, T.: Fortran Program for Calculating Transonic Velocities on a Blade-to-Blade Stream Surface of a Turbomachine. NASA TN D-5427, Sept. 1969.
3. Dodge, P. R.: A 3D Heat Transfer Analysis Program. AFAPL-TR-77-64, AiResearch Mfg. Co., Oct. 1977.
4. Denton, J. D.: An Improved Time Marching Method for Turbomachinery Flow Calculation. ASME Paper 82-GT-239, Apr. 1982.
5. Tabakoff, W.; and Wood, B.: Three Dimensional Flow Measurements in a Vaneless Radial Turbine Scroll. NASA CR-167994, Nov. 1982.
6. Goldman, L. J., and Seasholtz, R. G.: Comparison of Laser Anemometer Measurements and Theory in an Annular Turbine Cascade with Experimental Accuracy Determined by Parameter Estimation. NASA TM-82860, 1982.

C. TURBINE HEAT TRANSFER

1. Graham, R. W.: Fundamental Mechanisms that Influence the Estimate of Heat Transfer to Gas Turbine Blades. NASA TM-79128, Aug. 1979.
2. Winstanley, D. K.; Booth, T. C.; and Dunn, M. G.: The Predictability of Turbine Vane Convection Heat Transfer. AIAA Paper 81-1435, July 1981.
3. Mayle, R. E.; and Marziale, M.: Spanwise Mass Transfer Variations on a Cylinder in "Nominally" Uniform Crossflow. Heat Transfer 1982, vol. 6, Grigull E. Hahne, K. Stephan, and J. Straub, eds., Hemisphere Publ. Comp., 1982, pp. 135-140.
4. Liebert, C. H.; Gaugler, R. E.; and Gladden, H. J.: Measured and Calculated Wall Temperatures on Air-Cooled Turbine Vanes with Boundary Layer Transition. NASA TM-83030, 1982.
5. Stepka, F. S.; and Gaugler, R. E.: Comparison of Predicted and Experimental External Heat Transfer Around a Film Cooled Cylinder in Crossflow. NASA TM-83017, Nov. 1982.
6. Papell, S. S.; Wang, C. R.; and Graham, R. W.: Film-Cooling Effectiveness With Developing Coolant Flow Through Straight and Curved Tubular Passages. NASA TP-2062, Nov. 1982.
7. Yeh, F. and Stepka, F. S.: Review and Status of Heat Transfer Technology for Air Cooled Passages of Turbine Blades. NASA TP to be published.
8. Florschuetz, L. W.; Metzger, D. E.; and Truman, C. R.: Jet Array Impingement with Crossflow-Correlation of Streamwise Resolved Flow and Heat Transfer distribution. NASA CR-3373, Jan. 1981.
9. Morris, W. D.; and Ayhan, T.: Observations on the Influence of Rotation on Heat Transfer in the Coolant Channels of Gas Turbine Rotor Blades. Proc. Inst. of Mech. Engi., London, vol. 193, 1979, pp. 303-311.
10. Simon, T. W.; et al.: Turbulent Boundary Layer Heat Transfer Experiments: Convex Curvature Effects, Including Introduction and Recovery. (HMT-32, Stanford Univ.; NASA Contract NAG-3.) NASA CR-164619, Nov. 1980.
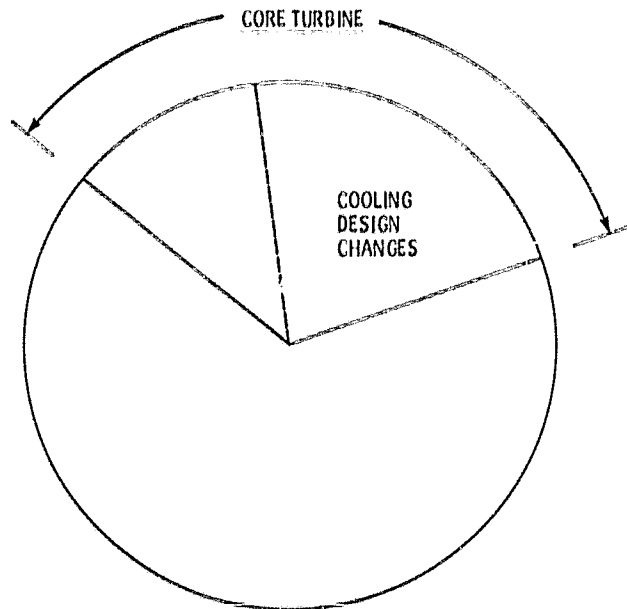
Figure 1. - New engine development cost: $500 M to 1200 M total (1979 dollars), 10-40% in core turbine (2/3 in fixes).

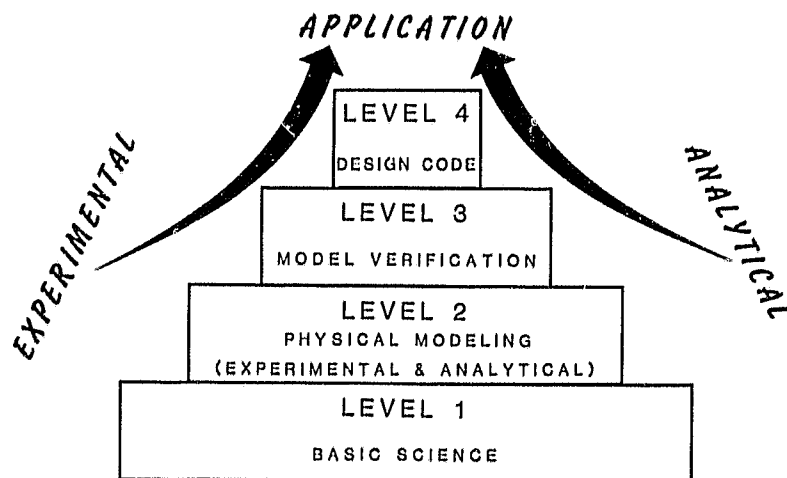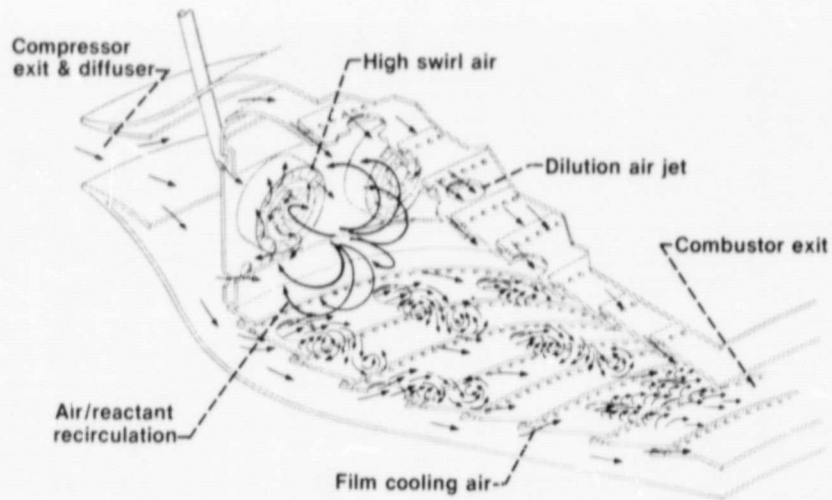Result: $30 M to 300 M in area of heat transfer research impact.

Figure 2. - Structure of design code evolution.

Compressor exit & diffuser

High swirl air

Dilution air jet

Combustor exit

Air/reactant recirculation

Film cooling air

- FULLY 3-DIMENSIONAL FLOW
- HIGH TURBULENCE LEVELS
- CHEMICAL REACTION/HEAT RELEASE
- 2 PHASE WITH VAPORIZATION

Figure 3. - Combustor flow phenomena.

RECEIVERS

TRANSMITTER

C-82-2247
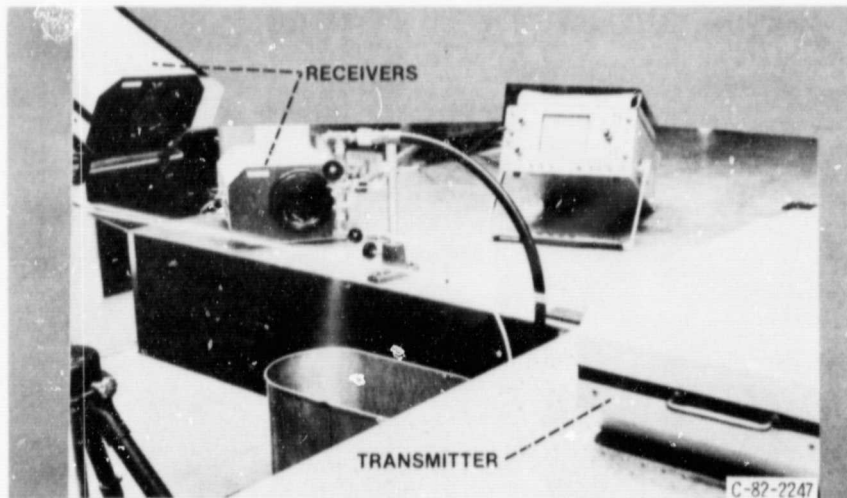
Figure 4. - DSI experiment configuration fuel nozzle spray characterization two-color two-component measurements.
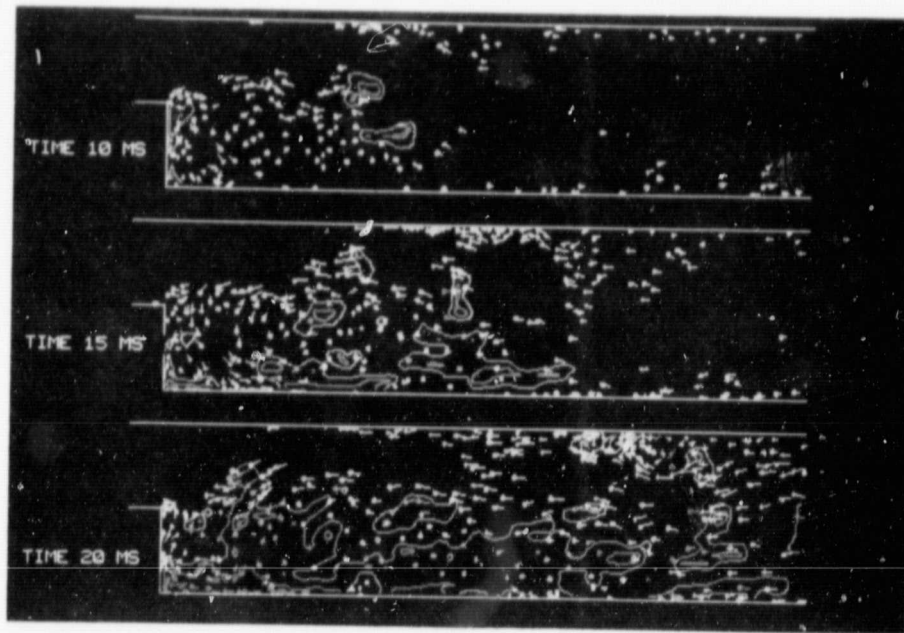
Figure 5. – Modeling interface motion of combustion reynolds number 10 000 (MIMOC).

TOP VIEW

$$THETA = (T_\infty - T)/(T_\infty - T_J)$$

X/H = 0.25   0.5   1   2
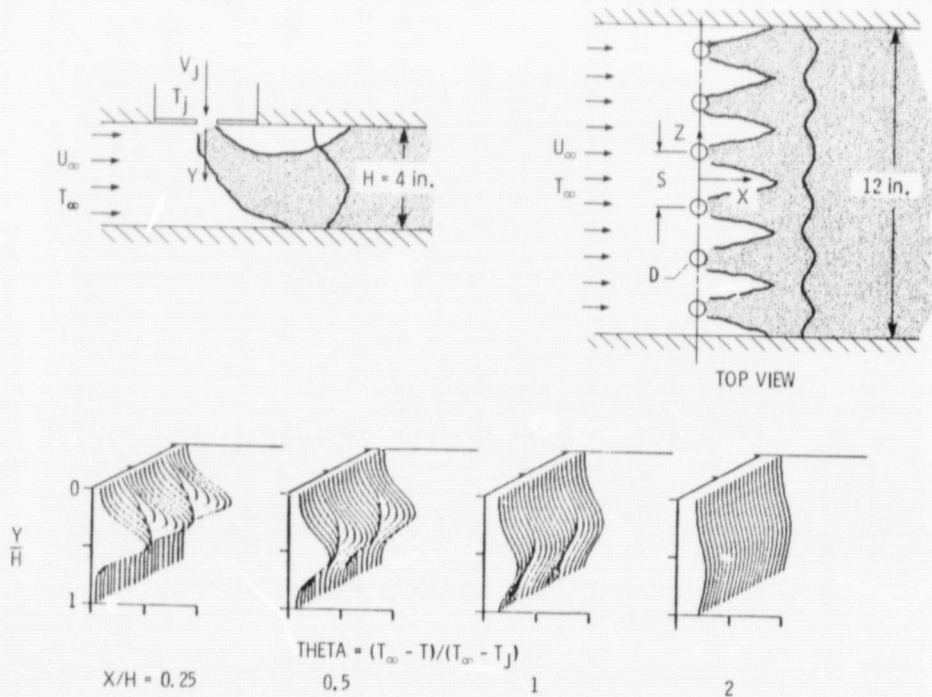
Figure 6. – Dilution jet mixing (S/H = 0.5, J = 32).
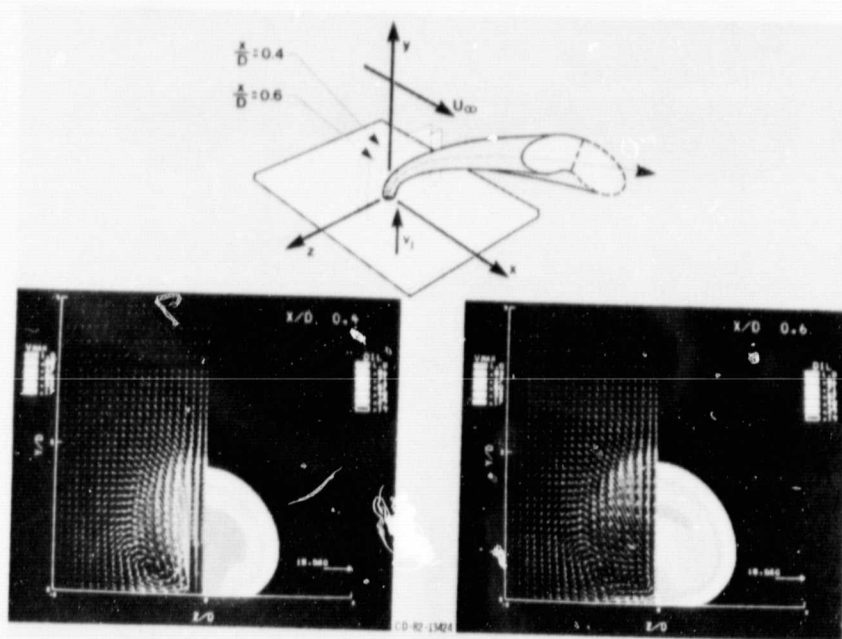
Figure 7. - Three dimensional dilution jet calculation.
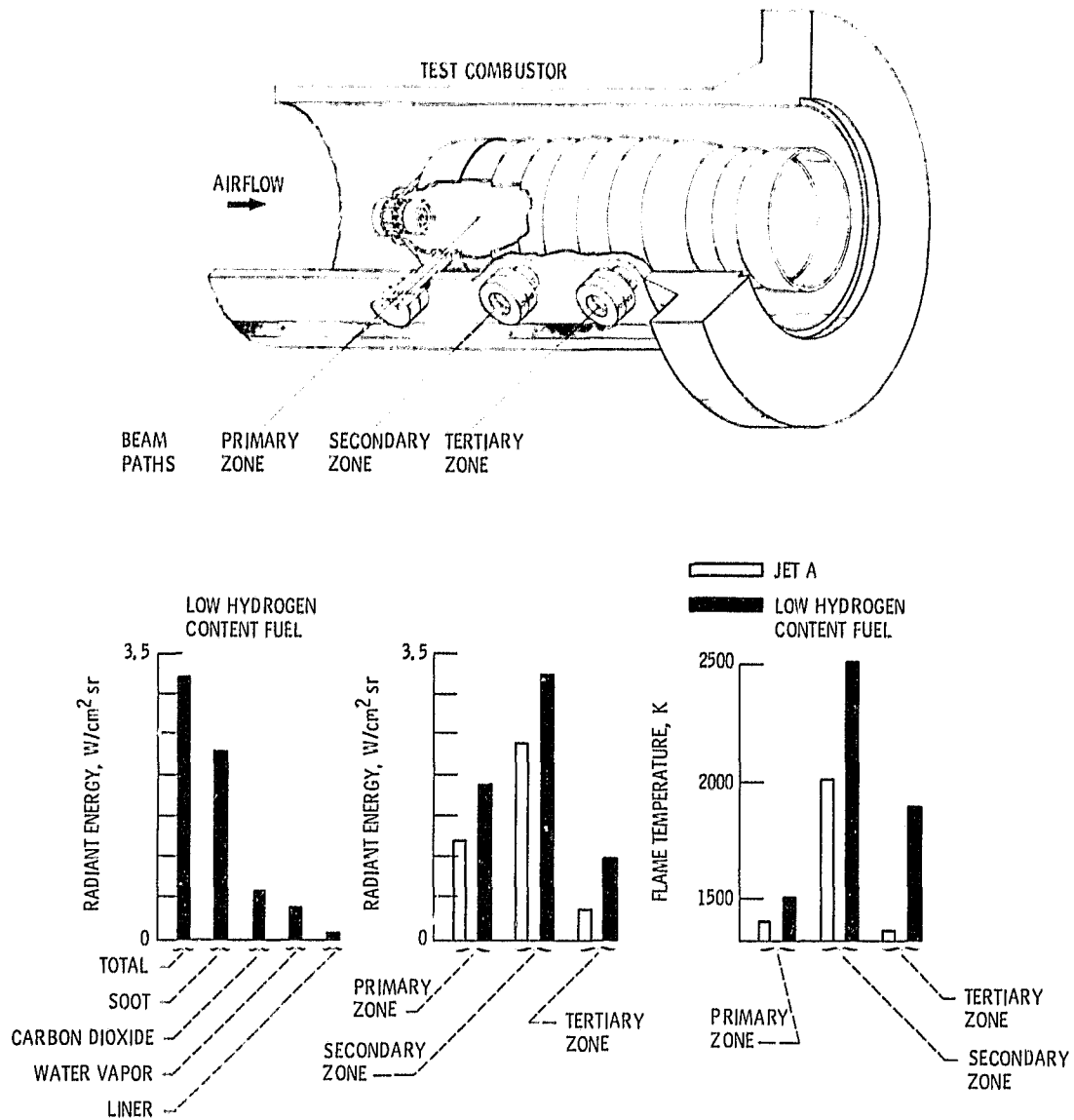
Figure 8. - Effect of fuel type on flame radiation.
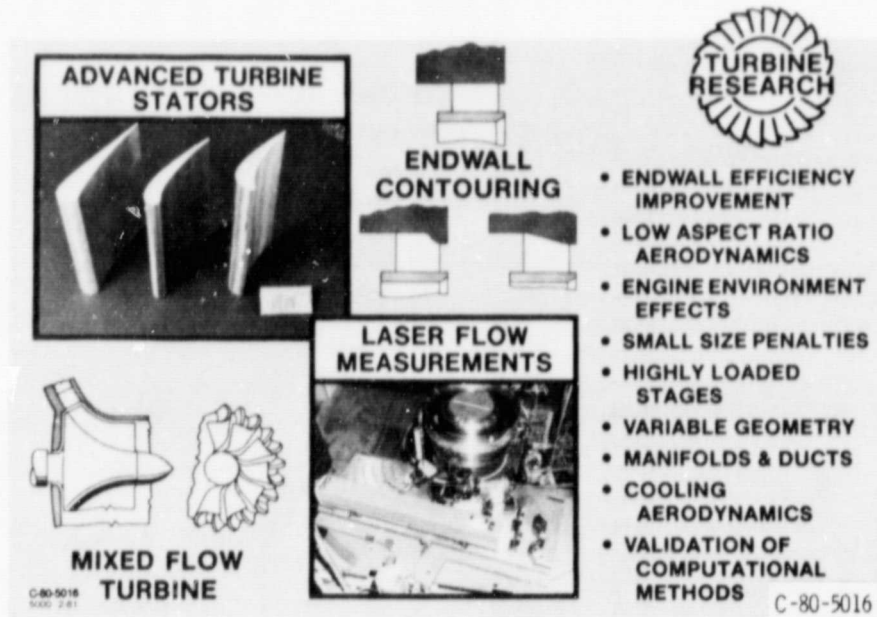
Figure 9. - Turbine aerodynamics R&T.
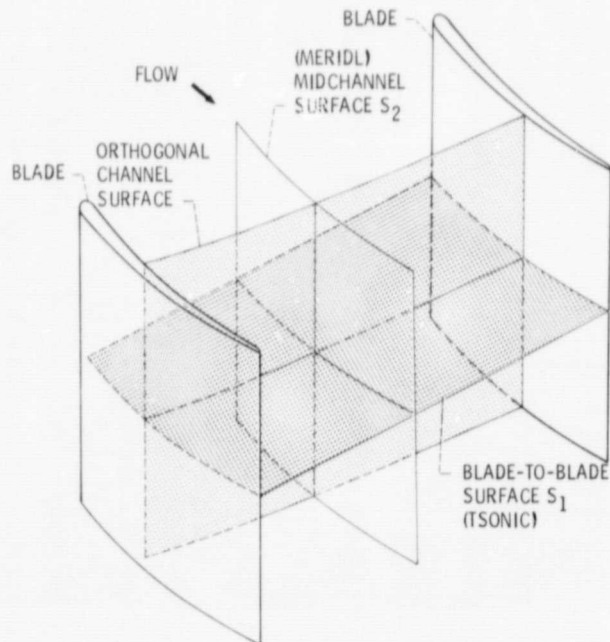
Figure 10. - Two-dimensional analysis surfaces in a turbomachine.

IMPACT:

PROVIDES VALIDATION OF THREE-DIMENSIONAL TURBOMACHINERY COMPUTER
PROGRAMS

EXPERIMENTAL APPARATUS:

SCOPE:

LASER MEASUREMENTS COMPARED WITH CALCULATIONS FROM
TWO INVISCID AND ONE VISCOUS 3-D COMPUTER CODES

RESULTS:

COMPARISON OF LASER MEASUREMENTS IN THE BLADE-TO-BLADE
PLANE WITH THEORY AT 80% AXIAL CHORD
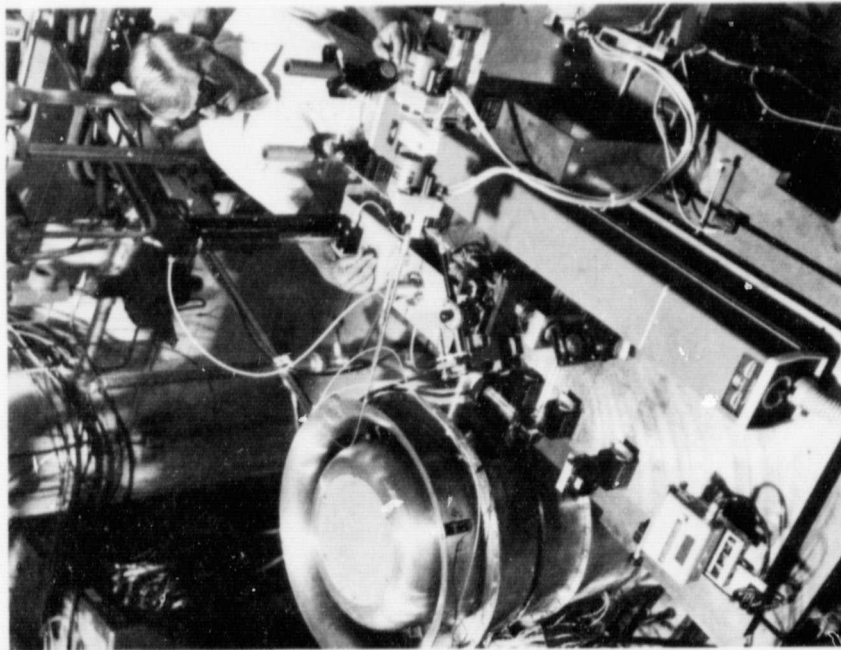
Figure 11. - Comparison of laser velocity measurements with theoretical predictions.
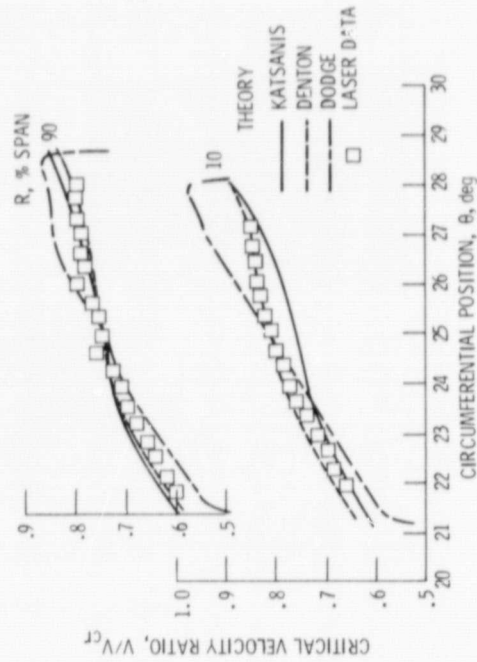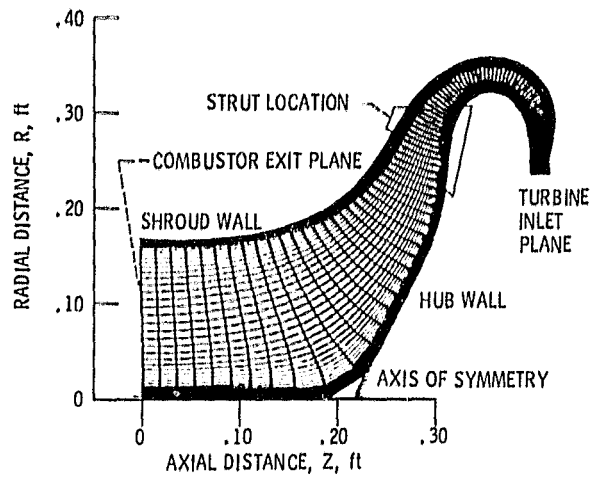
Figure 12. - Computational mesh for AGT101 turbine inlet duct.
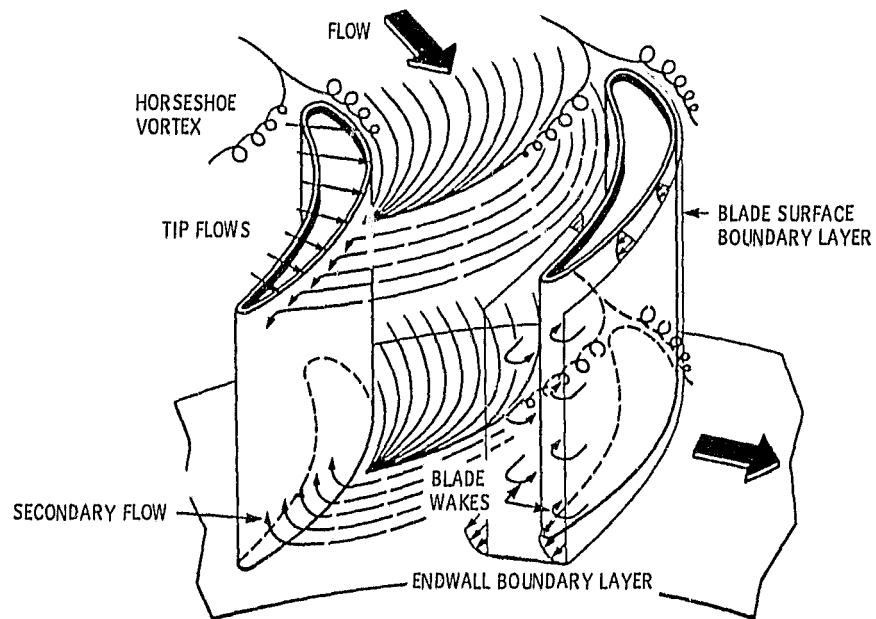
Figure 13. - Turbine blade row flow phenomena.
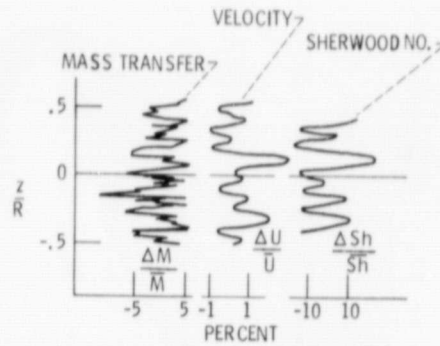
Figure 14. - New insight into stagnation
region heat transfer enhancement by
small disturbances. Correlation of
mass transfer, mean velocity and mesh
size. Sherwood number is mass
transfer equivalent of Nusselt number.
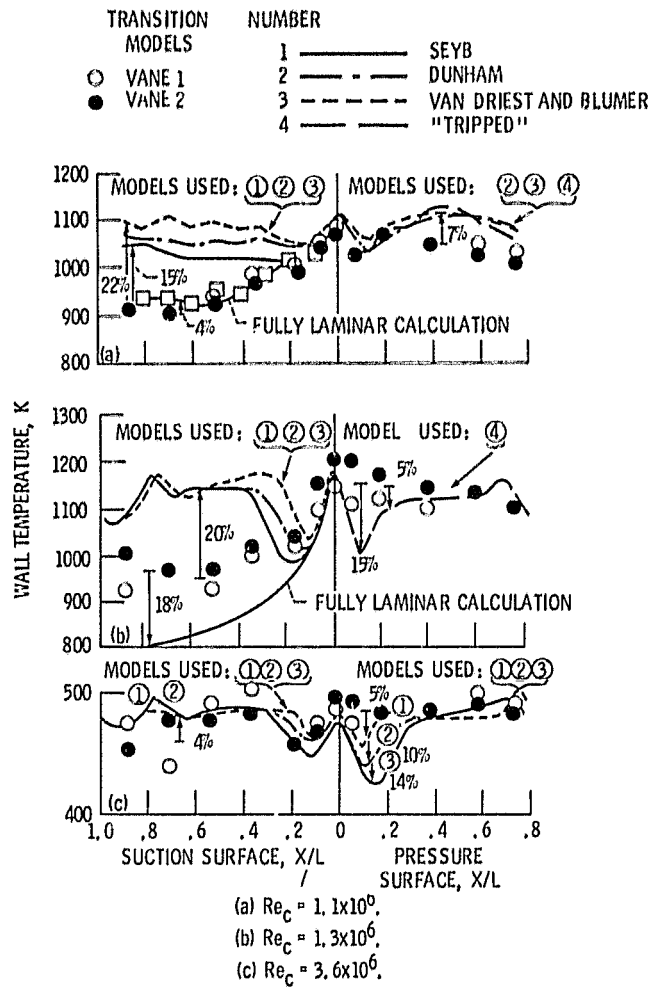

Figure 15. - Upstream turbulence effects tunnel.

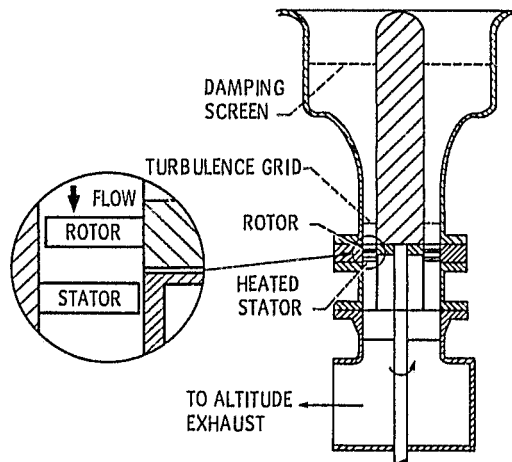Figure 16.- Comparison of analytical and measured
wall temperatures.

Figure 17. - Heat transfer in rotor wakes.

COOLANT

Figure 18. - Internal cooling.
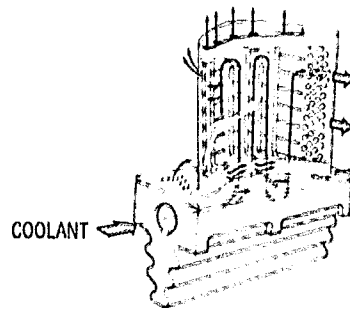

REGION 1
JET PLATE
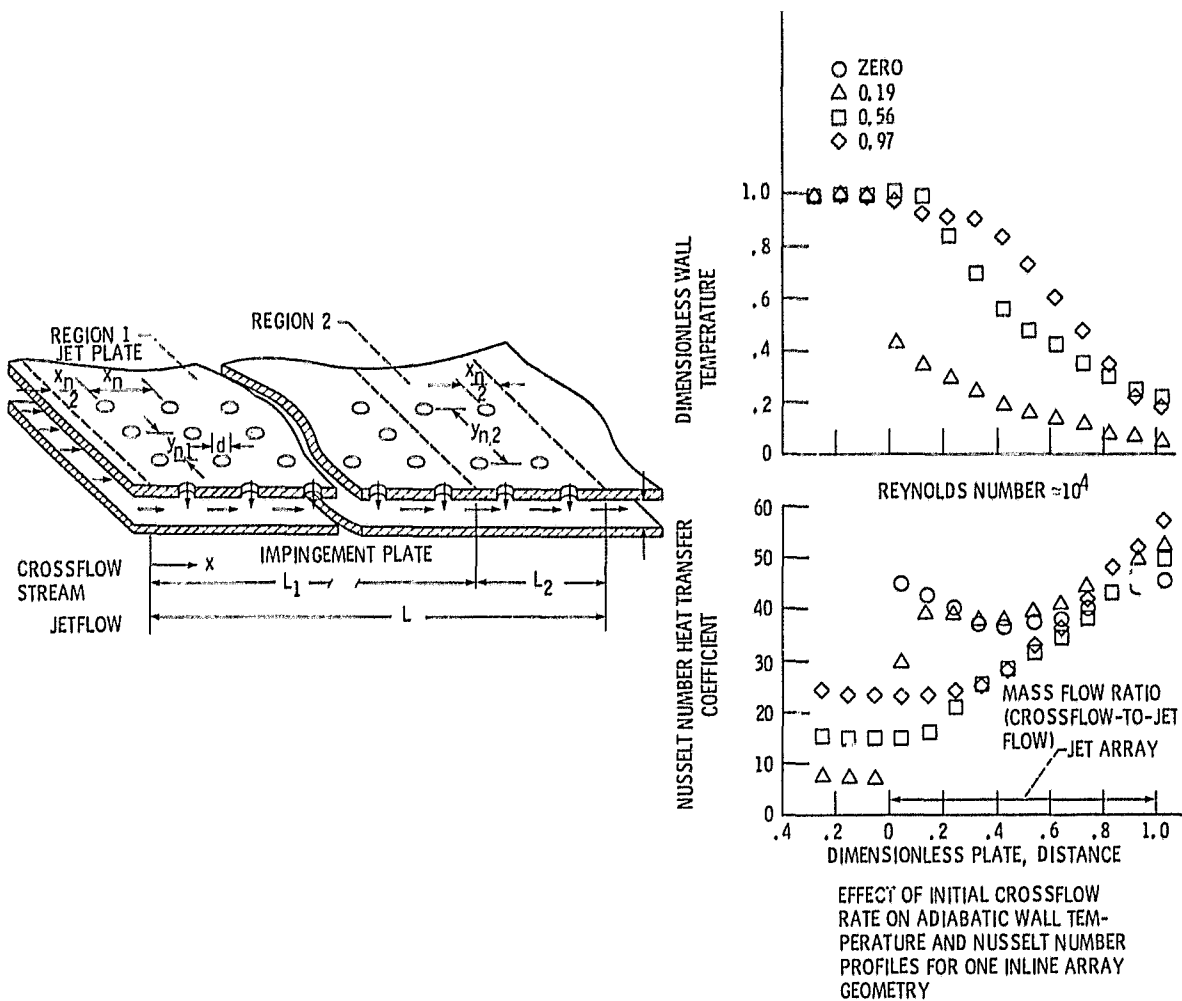
REGION 2

CROSSFLOW
STREAM

JETFLOW

IMPINGEMENT PLATE

○ ZERO
△ 0.19
□ 0.56
◇ 0.97

DIMENSIONLESS WALL TEMPERATURE

REYNOLDS NUMBER ≈10⁴

NUSSELT NUMBER HEAT TRANSFER COEFFICIENT

MASS FLOW RATIO
(CROSSFLOW-TO-JET
FLOW)

JET ARRAY

DIMENSIONLESS PLATE, DISTANCE

EFFECT OF INITIAL CROSSFLOW
RATE ON ADIABATIC WALL TEM-
PERATURE AND NUSSELT NUMBER
PROFILES FOR ONE INLINE ARRAY
GEOMETRY

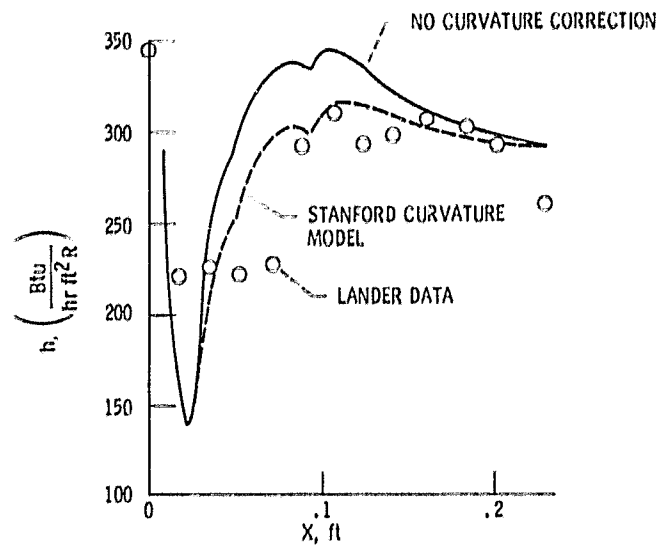Figure 19. - Impingement cooling heat transfer with realistic gas turbine type crossflow conditions, (ASU).

Figure 20. - Longitudinal curvature effect on turbulent boundary layer calculations; heat transfer coefficient vs surface distance from stagnation point; STAN5 calculation with and without Stanford Curvature model compared to data (Lander, 1969, airfoil 2, suction surface, Re = 4.87×10^5, Tu = 18%).
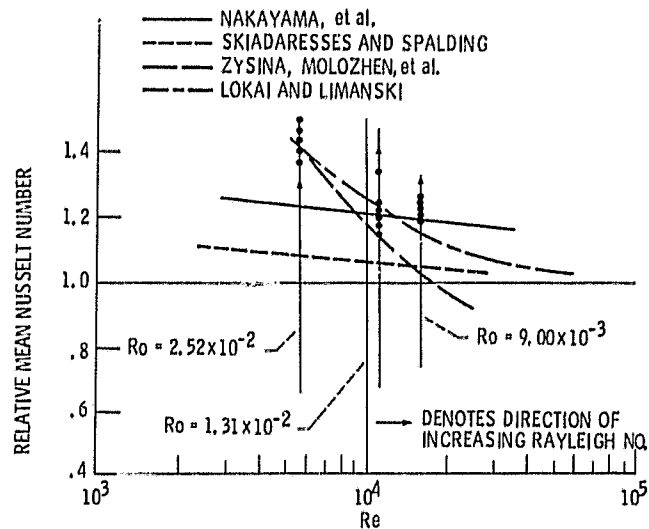

Figure 21. - Experimentally determined mean Nusselt numbers for a tube rotating about its longitudinal axis for radially inward flow. (Rotational speed = 1000 rpm. Nusselt numbers normalized to non-rotating values).
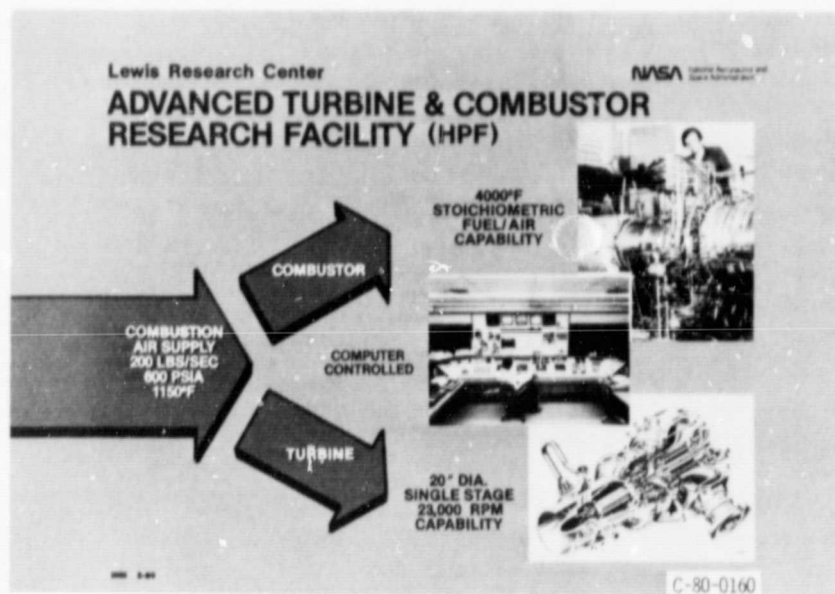
Figure 22. - Advanced turbine and combustor research facility (HPF).