NASA Technical Memorandum 83541

A Review of NASA Combustor and Turbine Heat Transfer Research

(NASA-TM-83541) A REVIEW OF NASA COMBUSTOR AND TURBINE HEAT TRANSFER RESEARCH (NASA) 26 p HC A03/MF A01 CSCL 21E

N84-14146

Unclas

G3/07 42814

Richard A. Rudey and Robert W. Graham
Lewis Research Center
Cleveland, Ohio

Prepared for the
Twenty-ninth Annual International Gas Turbine Conference
sponsored by the American Society of Mechanical Engineers
Amsterdam, The Netherlands, June 3-7, 1984

NASA

E-1912

A REVIEW OF NASA COMBUSTOR AND TURBINE HEAT TRANSFER RESEARCH

Richard A. Rudey and Robert W. Graham

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

SUMMARY

The thermal design of the combustor and turbine of a gas turbine engine poses a number of difficult heat transfer problems. In current designs, there is substantial evidence to indicate that the capability for estimating critical metal temperatures lacks the needed certainty for reliable design practice. Such uncertainty has cost much in terms of money and development time for the newer engines now installed in United States aircraft. The importance of improved prediction techniques becomes more critical in anticipation of future generations of gas turbine engines which will operate at higher cycle pressure and temperatures. Costly development modifications will have to be avoided by being able to predict metal temperatures more accurately.

Research which addresses many of the complex heat transfer processes holds promise for yielding significant improvements in prediction of metal temperatures. Such research involves several kinds of programs including: (1) basic experiments which delineate the fundamental flow and heat transfer phenomena that occur in the hot sections of the gas turbine but at low enthalpy conditions; (2) analytical modeling of these flow and heat transfer phenomena which result from the physical insights gained in experimental research; and (3) verification of advanced prediction techniques in facilities which operate near the real engine thermodynamic conditions.

In this paper, key elements of the NASA program which involves turbine and combustor heat transfer research will be described and discussed. Research conducted within the laboratories of the NASA Lewis Research Center and sponsored research being performed by universities and industrial laboratories will be included in this review.

INTRODUCTION

This report presents a summary of the mainstream effort in combustor and turbine heat transfer research being conducted and sponsored by the NASA Lewis Research Center (LeRC). The report focuses on the breadth of the heat transfer problems associated with the design and development of gas turbine engine hot sections and on the research, both currently underway and planned, that is producing the technological approaches to understand and solve these problems.

The contemporary gas turbine engine is a highly efficient and sophisticated piece of machinery. The rotating components are operating at very high aerodynamic efficiencies. To take advantage of these high efficiencies for reducing fuel consumption, designers are continually striving to increase engine thermal efficiency by increasing cycle pressure ratio and turbine inlet temperature. The gas generators of today's modern aircraft turbofan engines are operating at approximately 25:1 pressure ratios and 1600 K (2400° F) turbine inlet temperatures. Recent cycle study results, figure 1, indicate that continued fuel savings can be achieved by further increasing cycle pressure ratios up to as high as 64:1 and turbine inlet temperatures in excess of 1650 K (2500° F). These conditions impose formidable design conditions on the engine hot section. The primary hot section design problems that are being faced today and that must be faced in the future are related to maintaining acceptable reliability and life while taking advantage of the improved fuel efficiency that these cycle conditions offer.

Perhaps the biggest challenge facing today's hot section designer is trying to optimize the use of available cooling air in both the combustor and the turbine. To do this, he must be able to accurately predict the local metal temperatures that occur on the combustor liner and on the turbine blades and vanes. This requires the development of design codes which can accurately describe the hot gas environment, the coolant gas environment and the heat transfer coefficients that are associated with both the hot and cold surfaces of the materials used in the liners, blades and vanes. Today's designer is faced with a great deal of uncertainty in the prediction of metal temperatures, especially in the turbine (ref. 1). Metal temperature prediction uncertainties of up to 100 K (180° F) are commonly encountered. The end result of these uncertainties is that the designer must use an iterative process of design, experimental verification and redesign which increases development time and cost. Although this iterative process does reduce uncertainties, the results usually require "over-cooling" to insure that acceptable reliability and life are achieved and are generally nontransferable to new designs of differing configuration and operating conditions. The requirement for "overcooling" adversely affects the need to minimize the use of cooling air in order to maximize cycle efficiency, and the remaining uncertainty in metal temperature adversely affects the ability to accurately predict the life of the hot gas section materials.

The gas turbine heat transfer research program being conducted and sponsored by the LeRC is directly addressing the problem of metal temperature prediction uncertainty. The scope of the program encompasses fundamental research focused on developing improved understanding of the aerothermodynamic phenomena associated with the heat transfer process and applied research focused on the verification and application of improved aerothermodynamic models and design codes. The in-house research covers the spectrum from fundamental experiments to analytical modeling to "near-engine" environment experiments such as full-scale combustor and turbine component studies. Fundamental research into the effects of the aerodynamics and thermodynamics of the real gas environment on heat transfer and into the development of comprehensive analytical models of heat transfer phenomena are conducted under both university grant and research laboratory contracts. Experimental code verification and appropriate "benchmark" experiments are conducted using available research sources of the government, universities and industries. Much of the verification and application aspects of the research have been focused in a major NASA program called HOST (Hot Section Technology). The HOST program has been underway for several years under the direction of the LeRC. The primary goal of HOST is to produce improved design and analysis tools for predicting and designing gas turbine engine hot sections with improved reliability and durability.

This paper will summarize many of the ongoing Lewis research programs that are either complementary to, or an integral part of, the goal of evolving and developing more reliable and accurate heat transfer design tools for combustors and turbines. Highlights of the progress being made will be described and discussed.

CODE DEVELOPMENT METHODOLOGY

In the process of achieving an improved design procedure, there is an orderly evolvement of technical knowledge from basic fundamentals to design application. This process can be imagined to be a structure, as depicted in figure 2, made up of four levels representing building blocks of knowledge. The first level is basic science. Level 2 involves an interactive process of experimentation and analysis of discrete physical models. They simulate important physical phenomena in the combustor or turbine. The third level is an extension of the most relevant modeling of level 2. The parametric ranges of the experiments are extended to near-engine conditions and the analytical models further evaluated and modified. The peak of the structure is the incorporation of the pertinent knowledge flowing from the lower levels into a design code. This design code must be verified at experimental conditions representing the real engine environment.

The LeRC heat transfer research program for combustors and turbines has elements of all four of these levels. Progress is being made in each of the levels described and future challenges in heat transfer research that will affect the continued development of improved design codes and cooling concepts have been and are continuing to be identified.

COMBUSTOR HEAT TRANSFER

The combustion system of a modern gas turbine engine, figure 3, presents a very complex heat transfer problem in several regards. The hot gas environment is a complex phenomenon consisting of: (a) very turbulent aerodynamic flow fields; (b) varying levels of unsteady temperatures due to fuel injection, fuel reaction and dilution air mixing; and (c) the presence of varying heat radiation profiles emanating from gaseous and solid elements within the combustion zones. From a heat transfer point of view, the heat load that is tranmitted from this environment to the combustor liner also affects the cooling of the turbine blades.

Defining the combustion environment, including both the radiative and convection heat loads to the liner, is a major thrust in the LeRC's combustion research and technology base and HOST programs. Two primary factors that influence the level of heat load on the liner surface are combustor inlet air pressure and flame luminosity. A recent LeRC experiment (ref. 2) has yielded additional data regarding the impact of inlet pressure on both heat load and heat transfer coefficient, figures 4(a) and (b). These plots represent the total heat flux transmitted to the liner surface. Considerable effort is underway to determine the flame radiation contribution to the heat flux as it is effected by operating conditions, fuel properties and location within the combustor. The focus is not only to define the radiation contribution but to also evaluate the ability of current aerothermal combustion models to predict the level.

Another recent LeRC experiment was conducted (ref. 3) wherein a spectral radiometer was used to measure the radiant energy and temperature of the flame as a function of fuel hydrogen content and location within the combustor. In this experiment, the highest flame temperature and radiant energy was experienced within the secondary zone of the combustor. Significantly higher flame temperatures were obtained with a fuel having a hydrogen content about 2 percent lower than existing Jet A fuel. The primary contributor to the high radiant energy was soot contained within the combustion hot gases when using low hydrogen content fuel. This is expected because soot tends to form more readily as fuel hydrogen content is decreased. A comparison of a radiation prediction method (ref. 4) with experimental levels is shown on figure 5. Poor agreement was achieved in the secondary zone indicating the inability of the model to correctly predict conditions in the region where the highest radiant energy normally occurs. The resultant two-fold error in the predicted heat flux would present the liner designer with a problem of over-designing the cooling system if he used the subject model. This inability of today's aerothermal model to predict liner heat flux satisfactorily is one of the primary subjects of LeRC's HOST program.

Aerothermal Modeling

In response to current model inaccuracies, one of the present HOST activities is to assess the capability of existing aerothermal models to predict experimental results. Several major U.S. engine manufacturers are involved in this assessment. A key output of the assessment activity will be the definition of those elements of the available integrated aerothermal models that are the most critical to the predicted heat flux. Although this activity is in its early stages, several research efforts are providing an improved understanding of the combustion environment and the ability to predict the complex combustion flow field. A 2-D modeling program (ref. 5) has been developed to predict the temperature within a reacting flow field and the resultant qualitative comparisons are reasonably good, figure 6. In this model, random vortex dynamics are coupled with combustion kinetics to represent the time history of large-scale eddies that are formed in a 2-D combustor geometry with an abrupt step. Considerable future effort will be needed to extend this analysis to 3-D flow fields.

Further modeling activities are underway wherein the mixing of dilution jets with the mainstream reacting hot gases are being described (ref. 6). A typical computer graphics output from this analysis is shown in figure 7. This mixing process affects the temperature distribution both within the combustor (affecting the heat load to the liner) and at the combustor exit (affecting the heat load to the turbine). An interactive numerical model of the mixing process of multiple jets is being developed using dilution jet empirical mixing data (ref. 7). Typical temperature profile predictions are shown in figure 8. The model produces vectors which can be converted to colored graphics describing temperature profiles downstream of the dilution jet holes.

The progression of liner design practice from today's primarily empirically based methods to fully analytical techniques will surely require considerable time to develop and may never be completely practical to achieve. However, significant progress is being made to develop analytically based aerothermal combustion models that will reduce the amount of empiricism needed. This advancement will not only improve the accuracy of predicting combustor liner temperature, it will also have a substantial influence on reducing the experimental effort needed to develop acceptable liners for tomorrow's high performance gas turbine engines. Periodically, Lewis hosts conferences on its sponsored programs in combustion research which include liner heat transfer related activities. The latest conference was held in April 1984, and a conference proceeding similar to reference 8 will be available shortly after the conference.

Model Verification

One of the principal limitations to progress in model verification is the availability of diagnostic instrumentation and a comparative data base. Accurate, nonintrusive instrumentation is needed to measure local heat fluxes on combustor walls and liners. The radiation component is an increasingly important quantity that must be measured as combustion pressures increase and as the hydrogen content of fuels change. Efforts to analytically model the combustion process itself require measurement techniques that can distinguish the turbulent structure and scale and also identify the species in the various zones of the combustor.

Over the past decade considerable progress has been made in instrumentation methods that can be utilized in combustion research. Optical methods incorporating laser light have made possible sets of data not available before. Improved information about the fuel nozzle sprays, visualization of mixing processes, and diagnostics of combustion flame regions have given the researcher new data to work with in the process of evaluating models (levels 2 and 3 of fig. 1).

The applicability of analytical models to engine combustion conditions is being evaluated in a variety of full annular combustor geometries at inlet pressures up to 20 atmospheres in the LeRC High Pressure Facility. In addition, an experimental study to determine the influence of both the hot gas and cold gas side environment on liner heat transfer is underway via a contract to the General Electric Company. This experimental study is focused on determining how various combinations of combustion conditions and liner cooling concepts affect flame radiation and liner temperatures. The experimental apparatus, including the different liner cooling concepts, is shown in figure 9. Inlet air temperature, reference velocity and pressure will be varied for each liner concept along with parametric changes in combustion equivalence ratio and fuel hydrogen content. The range of test parameters will produce an empirical data base for developing and/or evaluating the predictive capability of combustor models.

TURBINE HEAT TRANSFER

A different thermal design challenge is confronted in the turbine than in the combustor. In several respects, the problem is more difficult. The rotation of the turbine rotor is a principal complicating factor which impacts the thermal and mechanical design. The geometry of the cooling system in the turbine is much more intricate than in the combustor. The local heating loads on the leading edge surfaces of the blades or vanes exceeds any local value in the combustor. In the turbine, one can identify most of the principal generic forced convection heat transfer research areas that are of current importance. Taken together, the complex geometry of the turbine and the diversity of heat transfer mechanisms involved result in a challenging design process.

Thermal design of a turbine blade is an iterative computation which involves estimates of the local heat input to the blade from the combustion gases, the effectiveness of cooling methods which utilize the coolant air bled from the compressor and the conduction of heat through the metal structure of the blade in question. The cooling methods and, in some designs, protective coatings are used to control the wall temperatures to an acceptable level.

Current design methods generally incorporate a two-dimensional numerical boundary layer analysis to compute the heat transfer from the hot gas into the blade. Such methods incorporate criteria for transition and may include empirical corrections for curvature and acceleration on eddy diffusivity formulations.

The numerical analysis of the heat transfer on the coolant side of the blade or vane wall is highly complicated due to the intricate coolant passage geometry and the variety of cooling methods that can be employed.

Prediction uncertainties which range from 84 to 110 K (150° to 200° F) are the state of the art today (ref. 1). Translated into blade life, this uncertainty represents one order of magnitude, which is intolerable.

Gas-Side Aerothermal Modeling

Estimating the heat transfer to the leading edge of vanes or blades is one of the major predictive uncertainties. At this stagnation region, the thermal and hydrodynamic boundary layers are initiated on the blade profile. Although much research has been devoted to similar flow geometries such as a cylinder in a cross flow, a number of fundamental questions relating to the turbine application remain unanswered. The influence of the upstream turbulence of the hot gas on the development of the blade boundary layer is an example. A two-dimensional wind tunnel which incorporates upstream rods or screens ahead of a cylindrical heat transfer model is being utilized to study this effect. The side walls are transparent so that visualization techniques can be used to observe the flow and heat transfer patterns. Simultaneous observations of both have resulted in photographs typified by figure 10. Temperature-sensitive liquid crystal sheets were wrapped around the cylinder. The scalloped profile of temperature which resulted from a disturbed upstream flow pattern is evident in the figure. Note also the stream tubes approaching the cylinder and the paired vortex patterns that develop at the leading edge.

Another important research topic is the transition of the boundary layer from laminar to turbulent. Almost all of the basic research in this topic has been devoted to the case of external flow aerodynamics applicable to aircraft. Over the past decade, major strides in understanding and analyzing this case have been achieved. For the analysis, small disturbance theory has been successfully applied in the prediction of transition because the boundary layer has not been subject to any major external disturbance. However, the turbomachinery environment is anything but quiescent; so the small disturbance approach is not applicable. Analytically, the large disturbance model is a much more difficult problem. The transition study community has labeled this "the large disturbance bypass path to transition." Serious attention to this problem is just beginning. At a recent LeRC hosted symposium on this subject, approximately 50 researchers discussed the transition phenomenon as it occurs in turbomachinery and made recommendations to guide the national research effort. One of the research programs reported on at this symposium was a NASA grant at Case-Western Reserve University. The initial objective of the work is to typify experimentally the spectrum of a large disturbance such as is found in turbomachinery flows as input information to the theory. The selection of the appropriate analytical method is a difficult problem and will be the focus of a long-term effort. As a short-term approach which provides guidance to the turbine designer, a more applied research approach reported on is being pursued at LeRC. The intent is to provide improved transition criteria which are employed in boundary layer models. The use of existing criteria (ref. 9) fail to depict the length characteristics of transition observed from heat transfer results in cascades. A return loop tunnel has been put into operation at The LeRC which promotes transition conditions for a developing boundary layer on the flat plate floor of the tunnel. A photograph of this tunnel is shown in figure 11. The test section where transition will occur is five feet long. Empirical representations of the length characteristics of transition derived from this experimental program will be inserted into boundary layer codes such as STAN 5.

Film cooling is taking on an increasingly important role in the cooling of advanced turbine blades. Consequently, it is the subject of continued research activity to assure that optimum design of the film cooling method is incorporated in blade designs. Without appropriate design guides, the amount of cooling may be inadequate; or in other cases, it can be excessive, resulting in turbine efficiency penalties. The generation of suitable design prediction guides for such a complex cooling process is a major challenge. At the LeRC, in-house and grant research in film cooling is being pursued. The in-house experimental research has centered on a fuller understanding of how the coolant flow conditions and geometry influence the effectiveness of film cooling (ref. 10).

NASA sponsored research at Stanford University is devoted to the effects of curvature in the main hot gas path on film cooling effectiveness. This work involves full coverage film cooling thus multiple cooling holes are involved. Detailed measurements of the boundary layer along a convex surface with and without film cooling have been made. The test section was carefully designed so that curvature only and not pressure gradient or acceleration effects are present in the two-dimensional duct. A diagram of the apparatus along with some heat transfer data for multiple row film cooling is shown in figure 12. Note the sharp discontinuity in the Stanton Number where the curvature begins. The objective of the overall program is to develop the necessary analytical modeling that can be imposed on boundary layer computational methods for prediction of wall temperatures with film cooling on convex or concave surfaces.

Gas-Side Verification

Much of the experimental work described thus far is two dimensional. The flow path through a turbine is highly complex; and, thus, knowledge of the three-dimensional properties of the flow are essential to the aerodynamic and thermal design of the turbine. Laser anemometer systems appear particularly suitable to detailed multi-dimensional measurements within blade passages because the detection is nonintrusive. At the LeRC, the laser anemometer is being utilized to measure three components of velocity at various flow locations within a blade passage. Usually the laser anemometer is capable of only two directions - both orthogonal to the optical axis. However, the Fabray-Perot interferometer technique has been applied to enable velocity measurements along the optical axis. These three-dimensional data results are being compared to three-dimensional computational codes. As reported in reference 11, the Denton inviscid code seems to do the best job of predicting the velocity data. Figure 13 shows a comparison of the data with the Denton code.

The Allison Gas Turbine Division of General Motors completed a comprehensive experimental study of heat transfer in a high temperature turbine cascade (ref. 12) This work was cosponsored by the Air Force and NASA. This extensive data set is being used to assess several analytical predictions including modifications to the STAN 5 boundary layer code and a Navier-Stokes Solver as a HOST project.

Coolant-Side Aerothermal Modeling

At the beginning of this section on thermal design, it was mentioned that the design process is an iteration between the estimates of the gas-side heat transfer and the coolant side. The internal geometry of a turbine vane or blade is very complicated. Representing the tortuous paths and cooling mechanisms is a formidable challenge. In the flow passages, the boundary layers are always in some state of development and are subject to several flow direction changes and strong body force components in the rotor. When compared to the familiar reference case of fully developed flow in a tube, the flow history in a multi-pass coolant tube is radically different; and, consequently, so is the heat transfer distribution. The results of an experimental simulation of flow conditions within a multi-pass coolant tube are shown in figure 14. The data show the significant differences between fully developed conditions and those typical of coolant passages for smooth and rough surfaces. Note the dramatic spikes in heat transfer that occur in the 180° bends of the coolant passage.

Under the HOST program, Pratt and Whitney Aircraft is conducting an experiment to examine the influence of buoyancy and coriolis forces on the forced convection heat transfer in a radial passage being rotated at high speed. The multi-pass geometry of the coolant passage is similar to the model being studied in the LeRC research program. Comparison of the results will enable conclusions about the nature of the coolant flow and the influence of rotation on multi-pass coolant flows.

One of the widely used cooling methods is impingement cooling. Jets of cooling air impinge on the back surface of a hot wall where augmented cooling is needed. Arizona State University conducted an extensive research program which examined the fundamental geometric parameters and the influence of cross flow on this type of cooling (ref. 13). The research showed that in-line hole geometry is preferable to staggered geometry for optimum cooling and that cross flow has a marked influence on the average heat transfer coefficient to the impingement-cooled plate.

Pin fins or pedestals are generally incorporated in the trailing edge of turbine blades. They serve as augmenters of the local heater transfer by turbulating the flow, and they also offer added rigidity to the structure of the blades. Their contribution to the effectiveness of cooling was not known accurately. A systematic heat transfer investigation of pin fin geometries was conducted. As shown in figure 15, the short L/D geometry of the pins exhibited a much lower heat transfer coefficient than long L/D rods referenced in the literature.

Most of the experimental modeling studies referred to in this section relate to the effects of geometry or rotational motion on local heat transfer coefficients. Comparisons have been made to predicted forced convection correlations available in the literature as a means of improving existing design codes. Undoubtedly, as more numerical or analytical methods become available, they will supplant the correlations as models of the internal cooling phenomena.

Coolant-Side Verification

Verification of the local coolant side heat transfer coefficients will be very difficult at nearengine conditions. The diminutive size of the coolant passages even for large turbines all but prohibits local measurements to be made. Consequently, it appears that the only verification at real engine conditions will be heat balances from measurements of the coolant bulk temperatures. The local heat transfer coefficient verification will have to come from the tests in simulated passage geometries at low heat flux rates.

CONCLUDING REMARKS

This paper presents a brief view of the NASA-sponsored research program in heat transfer that is directed toward the hot section of the gas turbine engine. It is important to recognize that this program involves the joint participation of industrial, university and NASA research capabilities. The cultivation of a strong infra-structure of these three elements is one of the major responsibilities of NASA. Technical meetings, such as this Gas Turbine Conference, serve as important forums where representatives of industry, university and government can exchange information on research findings. Effective communication is essential to technical advance.

In the INTRODUCTION, the need for the development of more reliable design procedures for the hot sections of the gas turbine was pointed out. The uncertainty in predicted metal temperatures must be reduced from its current estimated level of approximately 100 K (180° F). It is hoped that this uncertainty can eventually be brought down to 28 K (50° F). If achieved, this improvement would eliminate much of the expensive modification to fabricated components that make up such a large part of current engine development costs. Greater certainty in metal temperature prediction will result in longer life for the highly stressed hot components of an engine. In addition, improved certainty will enable more economical allocation of cooling air quantities in both the combustor and the turbine. The overall cycle efficiency of the engine will be improved by minimizing the cooling air budget.

As described in figure 1, the process of improving and certifying a thermal design methodology for the turbine or combustor involves a number of interdependent steps and orderly evolvement. From this process, design methods will become available that enable more confidence in initial designs done on paper and reduce costly trial and error modifications of actual engine hardware. Also as a result of this stronger interaction of research activity with the design process, a significant advancement in generic research will result which will contribute to the thermal engineering practice in applications outside of the gas turbine arena.

REFERENCES

1. Stepka, F. S., "Analysis of Uncertainties in Turbine Metal Temperature Prediction," NASA TP-1593, 1980.

2. Claus, R. W., Neely, G. M., and Humenik, F. M., "Flame Radiation and Liner Heat Transfer in a Tubular-Can Combustor", AIAA Paper 84-0443, Jan. 1984.

3. Humenik, F. M., Claus, R. W., and Neely, G. M., "Parametric Study of Flame Radiation Characteristics of a Tubular-Can Combustor," NASA TM-83436, 1983.

4. Srivatsa, S. K., "Computations of Soot and NO_x Emissions From Gas Turbine Combustors," Rept. 21-4309, Garrett Turbine Engine Co., Phoeniz, Ariz., May 1982. (NASA CR-167930.)

5. Ghoniem, A. F., Marek, C. J., and Oppenheim, A. K., "Modelling Interface Motion of Combustion (MIMOC). A Computer Code for Two-Dimensional, Unsteady Turbulent Combustion," NASA TP-2132, 1983.

6. Claus, Russell W., "Calculation of a Single Jet in Crossflow and Comparison With Experiment," AIAA Paper 83-0238, Jan. 1983.

7. Holdeman, J. D., "Perspectives on the Mixing of a Row of Jets With a Confined Crossflow," AIAA Paper 83-1200. June 1983.

8. Combustion Fundamentals Research, NASA CP-2268, 1983.

9. Gaugler, R. E., "Some Modifications to, and Operational Experiences With, the Two-Dimensional Finite Difference Boundary Layer Code, Stan 5," ASME Paper 81-GT-88, Mar. 1981.

10. 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, 1982.

11. 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.

12. Hylton, L. D., et al., "Experimental Investigation of Turbine Endwall Heat Transfer," DDA-EDR-10363, General Motors Detroit Diesel Allison Division, Indianapolis, Ind., 3 volumes, Aug. 1981. (AD-A110332, AD-A110333, AD-A110334.)

13. Florschuetz, L. W., et al., "Jet Array Impingement Flow Distributions and Heat Transfer Characteristics. Effects of Initial Crossflow and Nonuniform Array Geometry," NASA CR-3630, 1982.

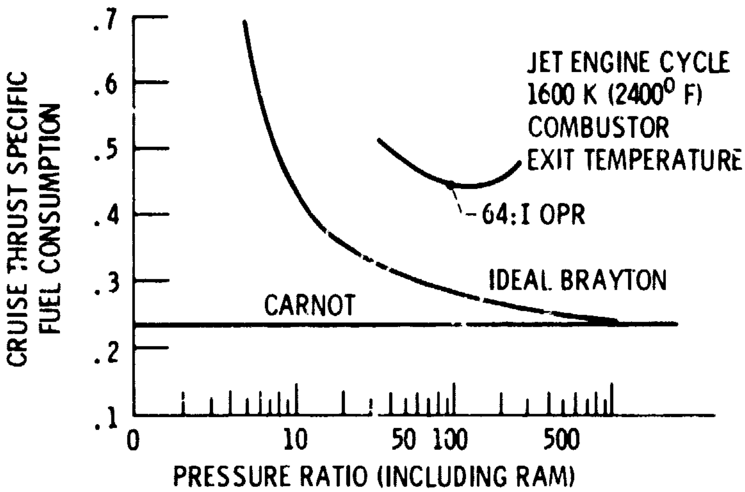


Figure 1. - Estimated optimum pressure ratio for jet engine cycle.

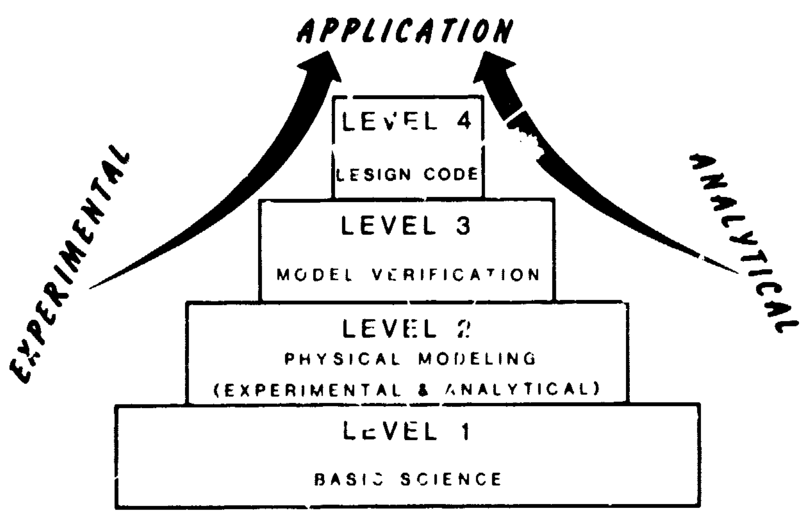
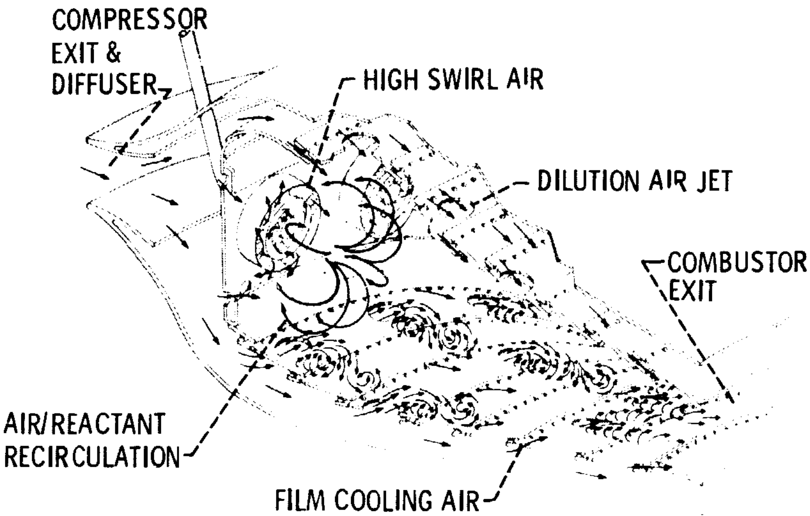


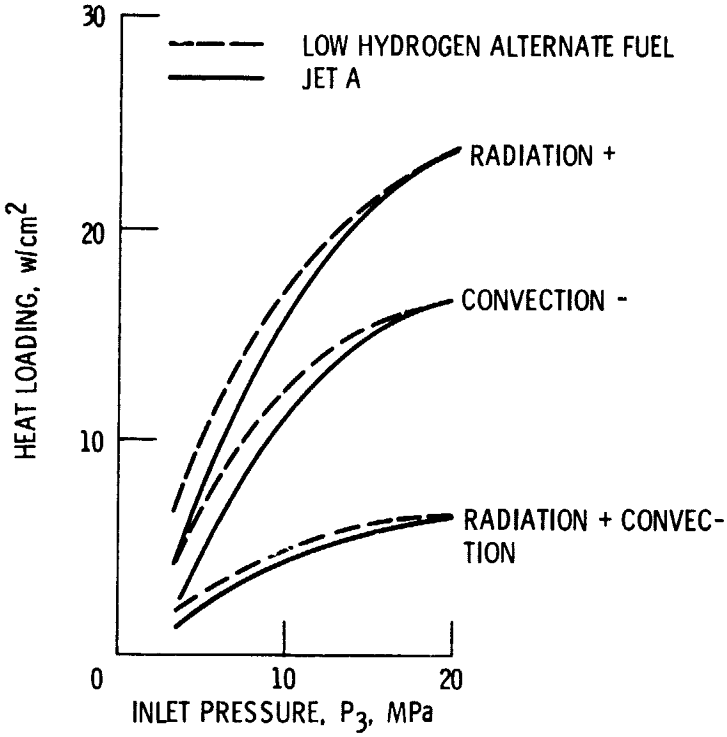
Figure 2. - Structure of design code evolution.

ORIGINAL PAGE IS
OF POOR QUALITY

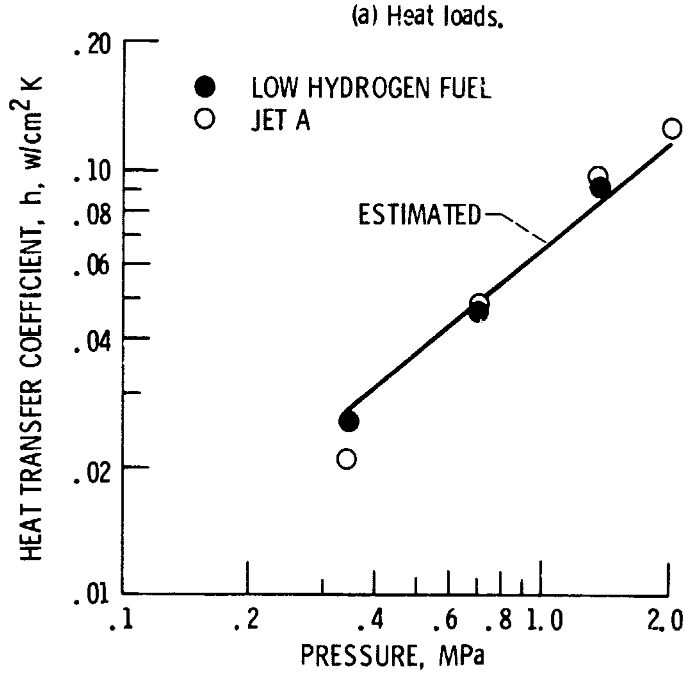


- FULLY 3-DIMENSIONAL FLOW
- HIGH TURBULENCE LEVELS
- CHEMICAL REACTION/HEAT RELEASE
- 2 PHASE WITH VAPORIZATION

Figure 3. - Combustor flow phenomena.



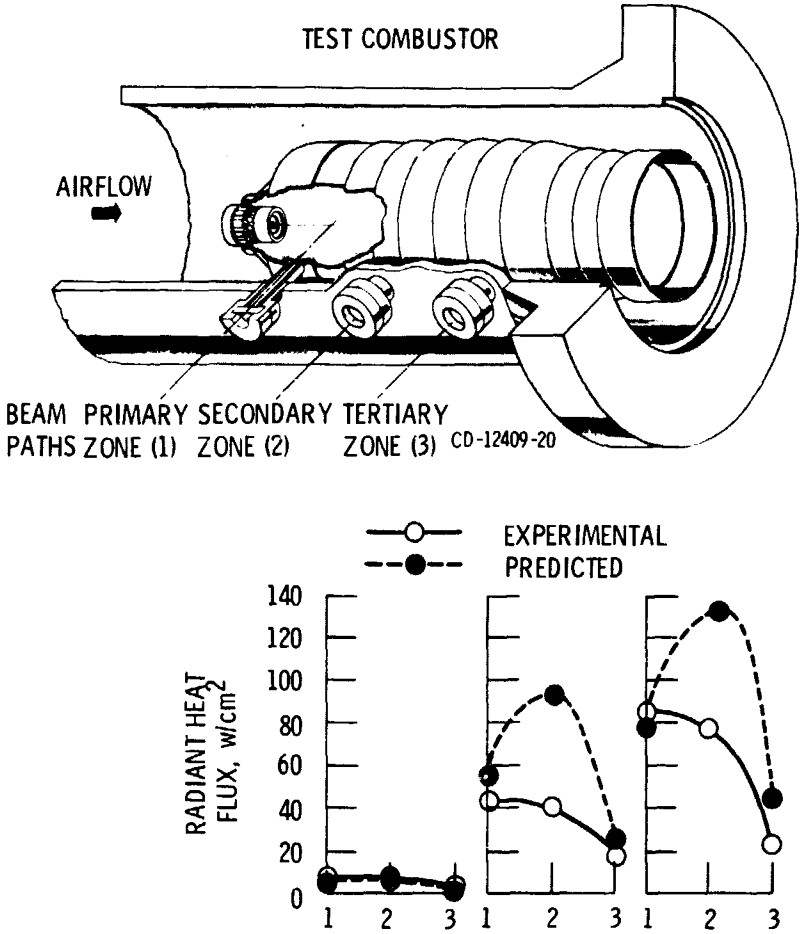
(a) Heat loads.



(b) Heat transfer coefficient.

Figure 4. - Calculated heat loads and heat transfer coefficients, h, as a function of inlet air pressure, P_3.

ORIGINAL PAGE IS OF POOR QUALITY



	IDLE	ZONE CRUISE	TAKEOFF
FUEL/AIR RATIO, F/A	0.007	0.014	0.018
INLET TEMPERATURE, T_3	400 K	620 K	715 K
INLET PRESSURE, P_3	269 kPa	700 kPa	1740 kPa

Figure 5. - Comparison of estimated radiant heat load with spectral measurements.

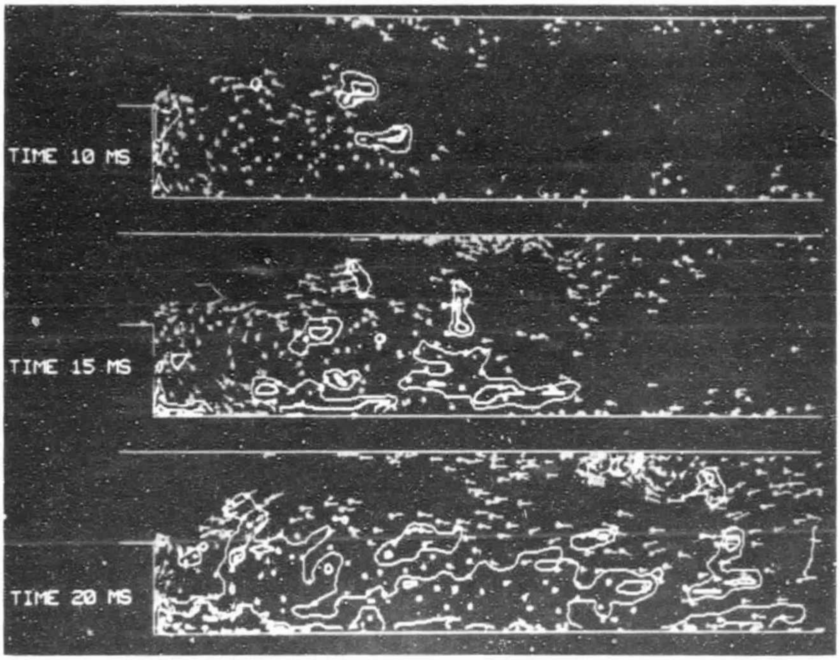


Figure 6. - Computational graphical representation of combustion process downstream of a 2D step. 3 time intervals.

CD-83-13778

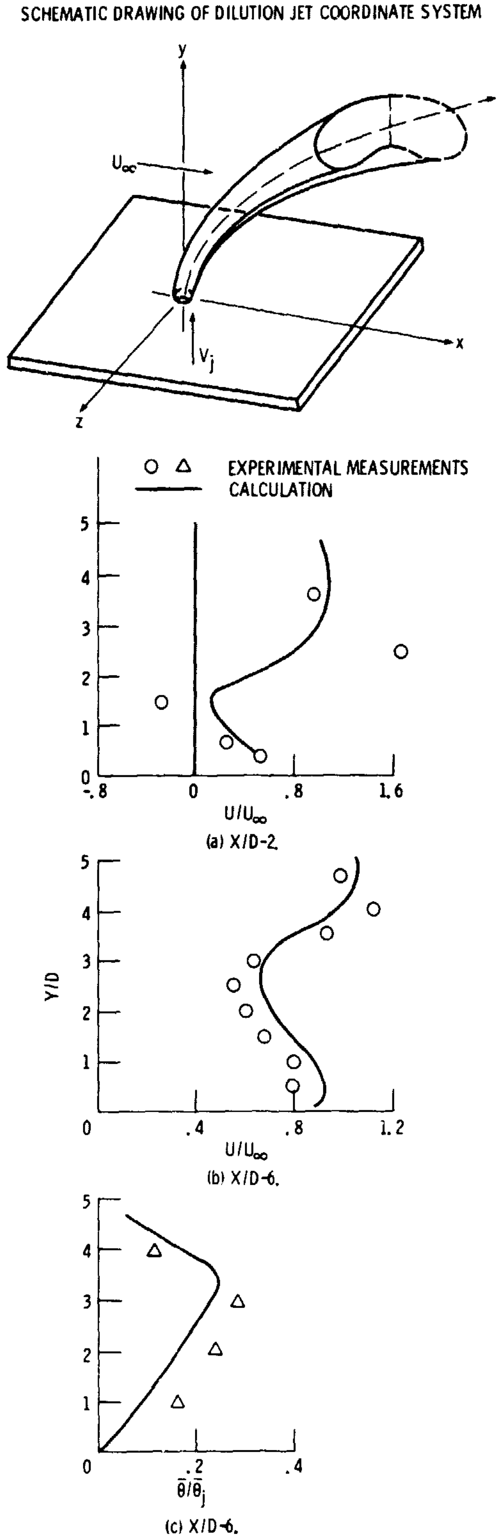


Figure 7. - Coarse grid (20x20x12) calculation of a single, free jet. Velocity profiles at (a) X/D-2, (b) X/D-6, and (c) normalized jet fluid concentration profiles at X/D-6. All profiles shown are through the jet centerline.

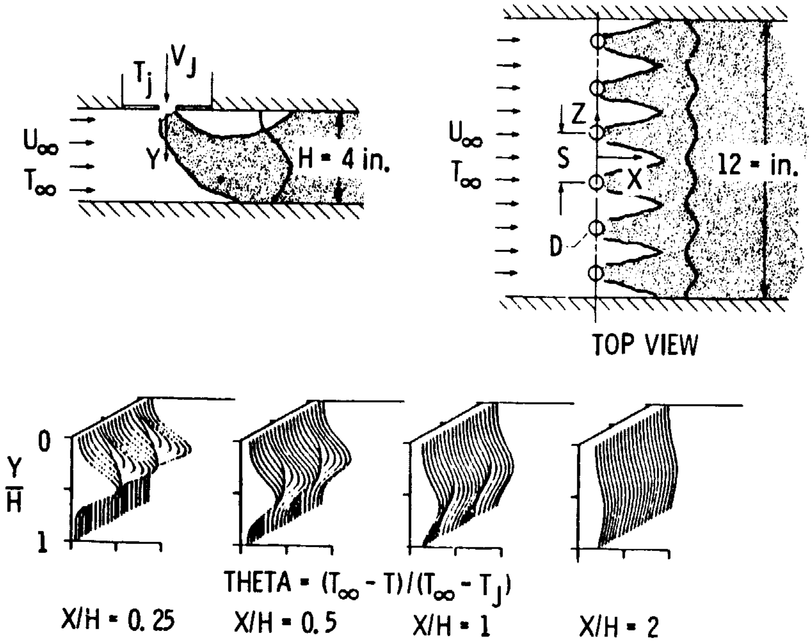


Figure 8. - Dilution jet mixing (S/H = 0.5, J = 32). Multiple jet numerical interaction approach. (J is dynamic P of jet to main stream.)

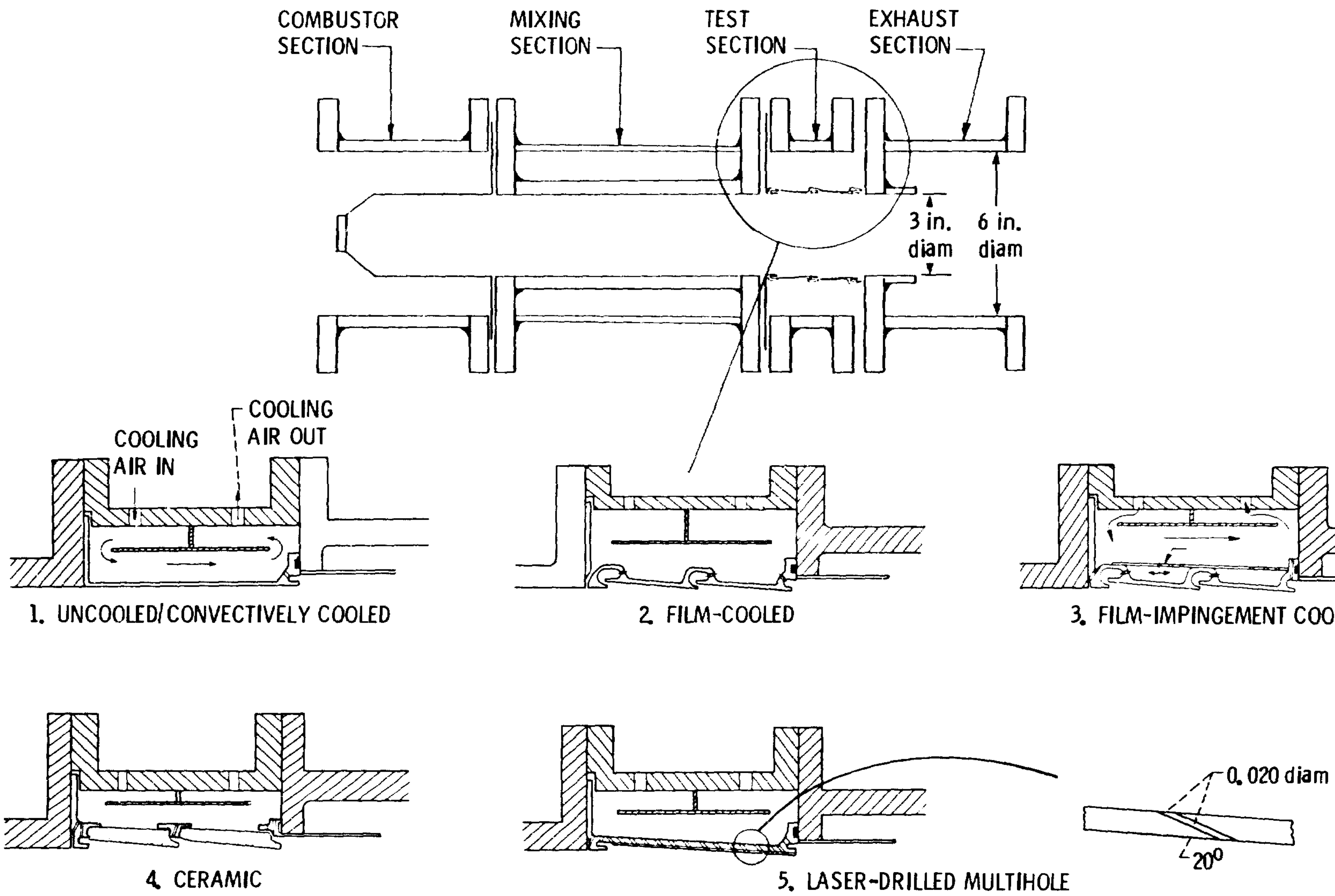


Figure 9. - Liner environment effects study.

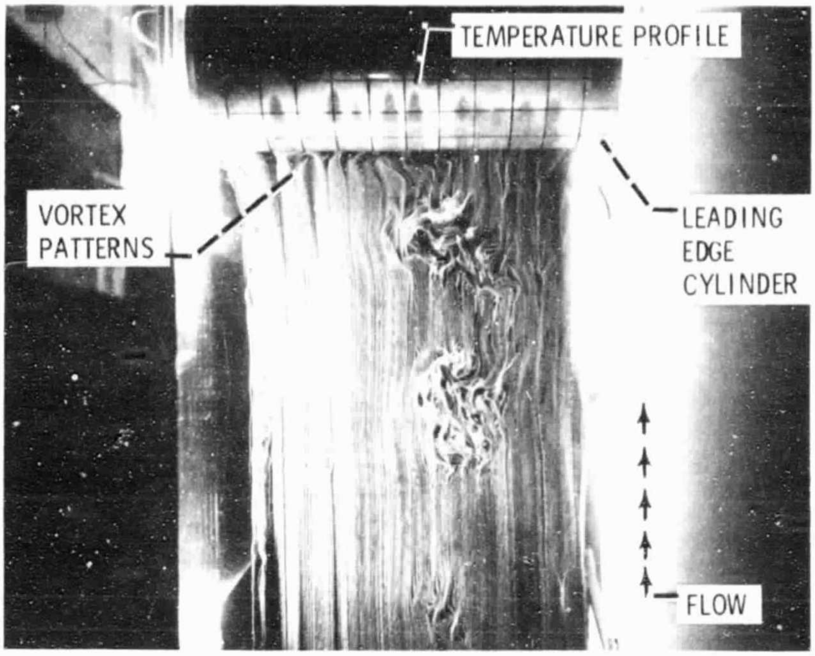


Figure 10. - Flow and thermal patterns visualization on a heated cylinder in cross flow.

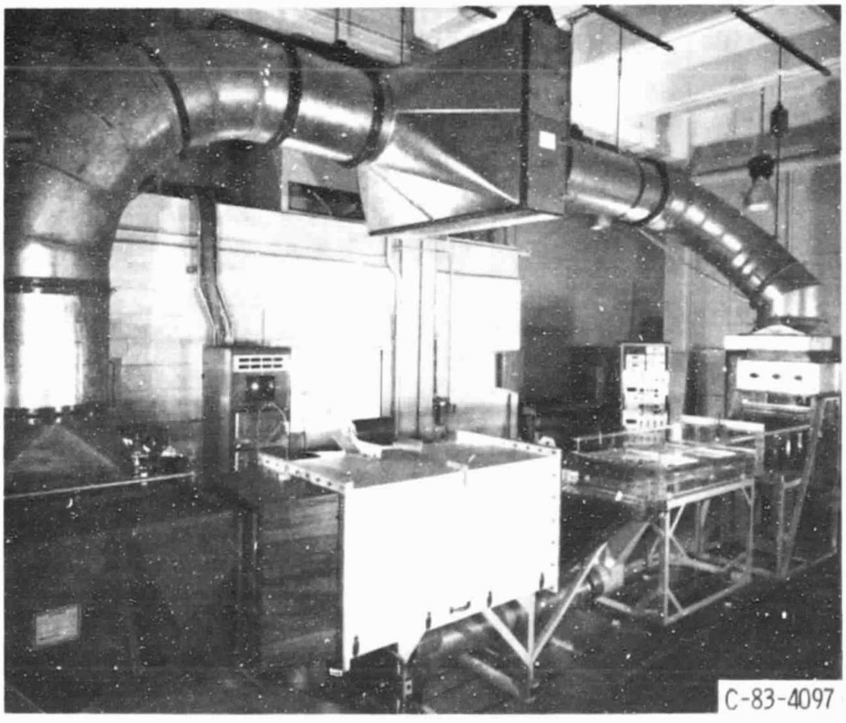


Figure 11. - Lewis transition tunnel.

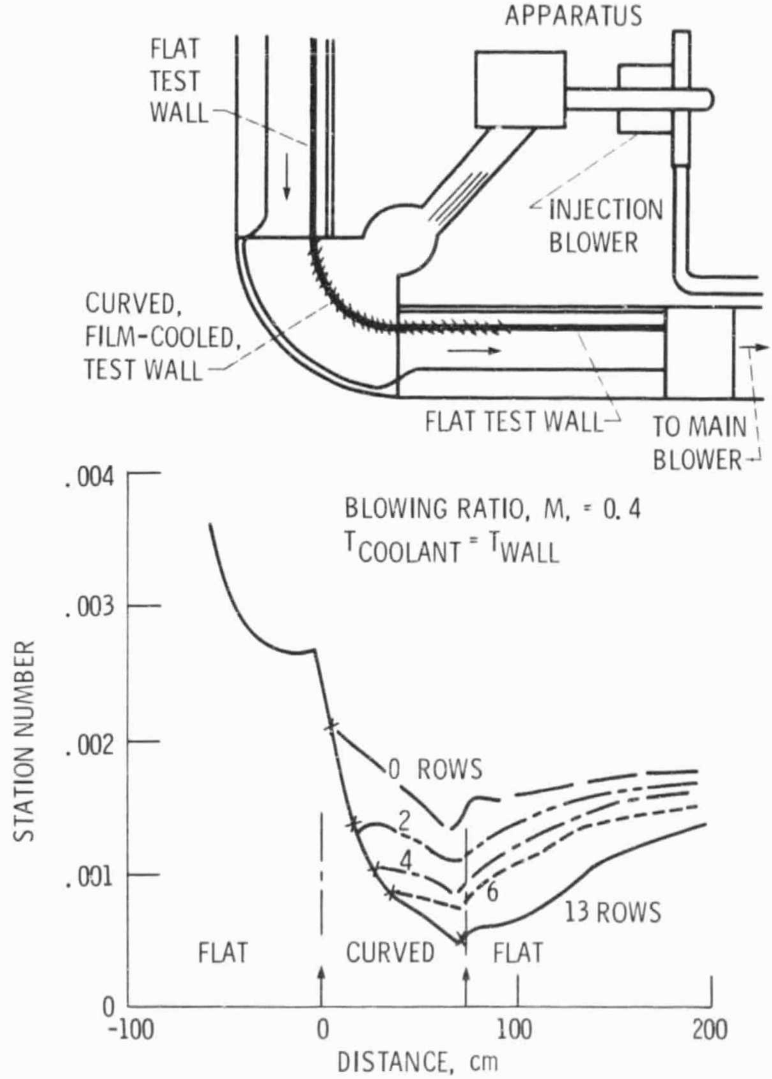
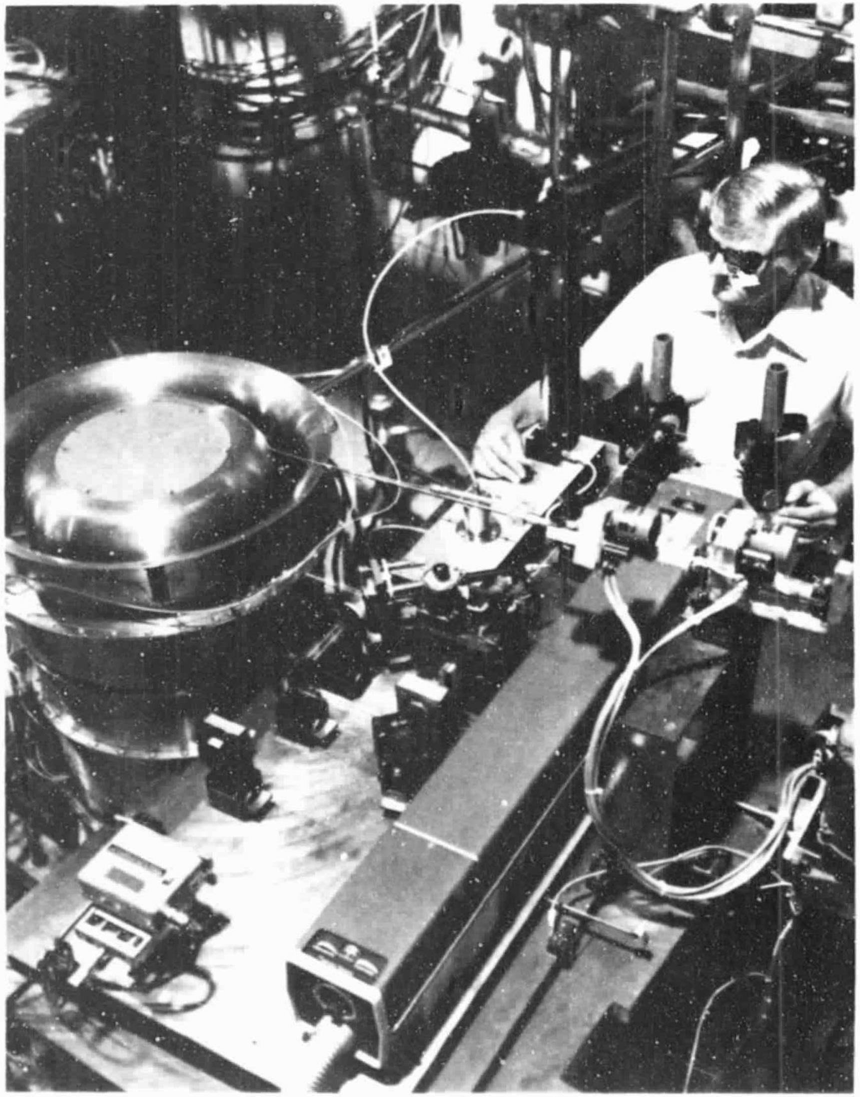


Figure 12. - The effect of curvature and multiple rows of holes on film cooling.

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

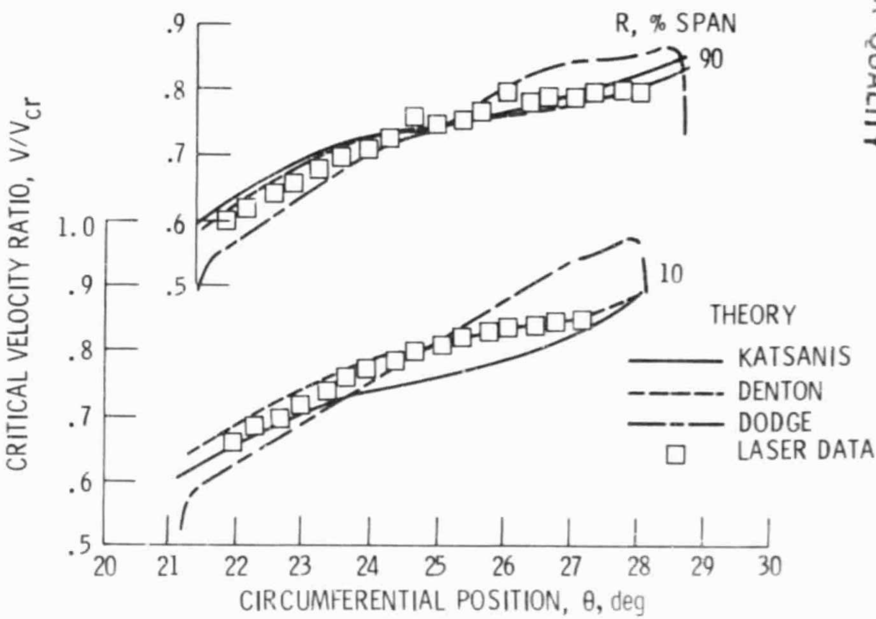


Figure 13. - Comparison of laser velocity measurements with theoretical predictions.

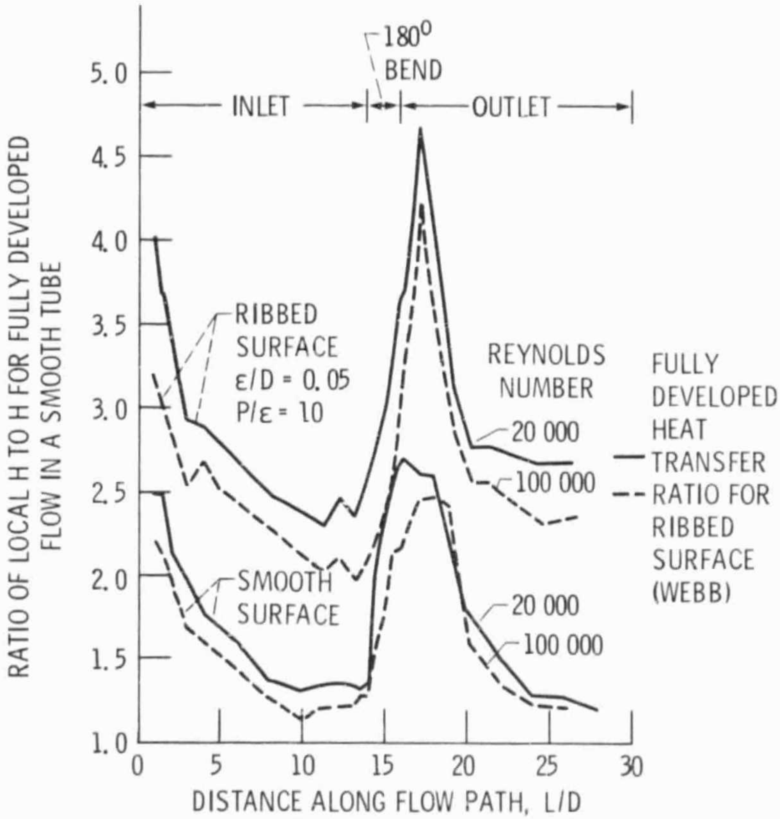


Figure 14. - Heat transfer in square duct with U-shaped bend in an internal cooling passage.

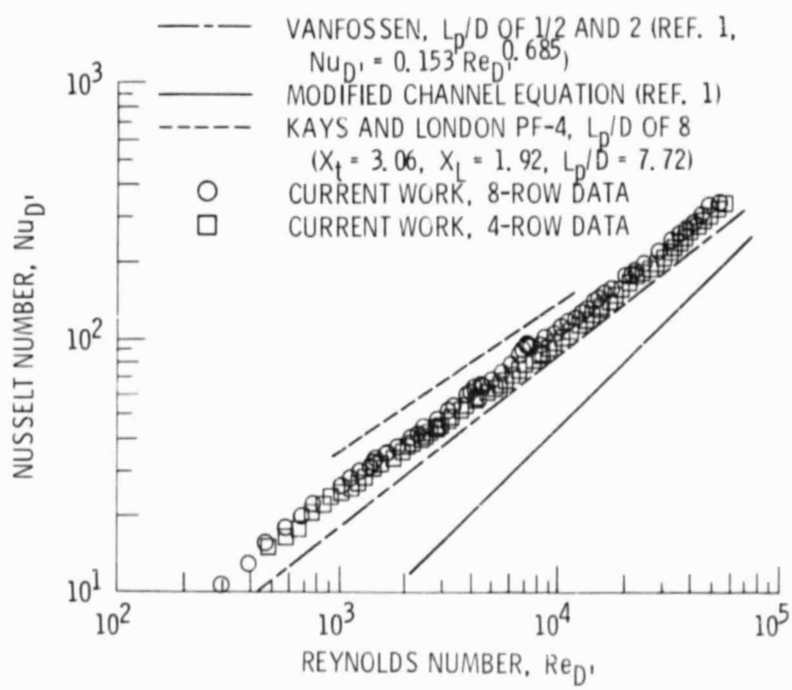


Figure 15. - PIN FIN geometry heat transfer compared to large L/D and channel geometries.