

52-07
N88-15792 117865
23P

TURBOMACHINERY

Robert J. Simoneau, Anthony J. Strazisar,
Peter M. Sockol, Lonnie Reid,
and John J. Adamczyk

ABSTRACT

The discipline research in turbomachinery, which is directed toward building the tools we need to understand such a complex flow phenomenon, is based on the fact that flow in turbomachinery is fundamentally unsteady or time dependent. Success in building a reliable inventory of analytic and experimental tools will depend on how we treat time and time-averages, as well as how we treat space and space-averages.

E-3798-0

The raw tools at our disposal - both experimental and computational - are truly powerful and their numbers are growing at a staggering pace. As a result of this power, a case can be made that we are currently in a situation where information is outstripping understanding. The challenge is to develop a set of computational and experimental tools which genuinely increase our understanding of the fluid flow and heat transfer in a turbomachine.

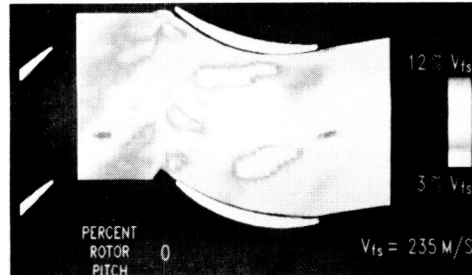
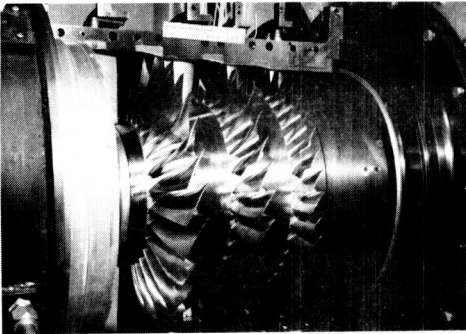
The following viewgraphs outline a philosophy based on working on a stairstep hierarchy of mathematical and experimental complexity to build a system of tools, which enable one to aggressively design the turbomachinery of the next century. Examples of the types of computational and experimental tools under current development at Lewis, with progress to date, are examined. The examples include work in both the time-resolved and time-averaged domains.

Finally, an attempt is made to identify the proper place for Lewis in this continuum of research.

TURBOMACHINERY RESEARCH PROGRAM

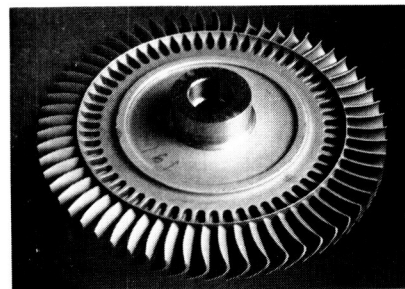
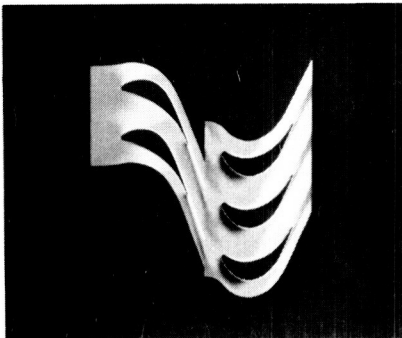
The long-range goal of the turbomachinery research program at NASA Lewis is to establish a validated three-dimensional viscous analysis capability for multistage turbomachinery, including unsteady effects and surface heat transfer. A full range of experimental and computational tools are being brought to bear on this problem in order to genuinely increase our understanding of the fluid flow and heat transfer in a turbomachine. The key to this understanding is the selection and successful application of the right tools for the right job.

TURBOMACHINERY RESEARCH PROGRAM



LONG-RANGE GOAL

ESTABLISH A VALIDATED 3D VISCOUS ANALYSIS CAPABILITY FOR MULTISTAGE TURBOMACHINERY, INCLUDING UNSTEADY EFFECTS AND SURFACE HEAT TRANSFER



CD-87-28977

ORIGINAL PAGE IS
OF POOR QUALITY

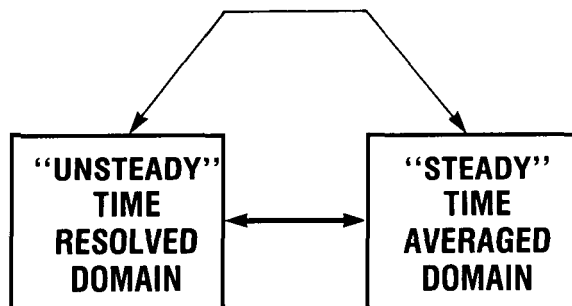
THEME

The theme of this presentation is that everything in a turbomachine is fundamentally unsteady (i.e., time dependent). This time dependency is further complicated by strong random disturbances. We have a choice. We can work in the time domain, which is expensive and time consuming; or we can work in the time-averaged domain, which is cheaper but yields less information. Furthermore, it is not simply an on/off, unsteady/steady situation. The averaging is by steps. Some of the time dependent nature can be averaged out, while some can remain. It is a matter of the time scale (or for that matter the length scale) over which the averaging is performed. Time average does not necessarily mean time independent or "steady". Thus, the question becomes both when and when not to average, and also how to average. A proper balance is required. Ultimately the engineer/designer is most interested in average information, but to get there one must properly handle time.

THEME

**EVERYTHING IN A TURBOMACHINE IS UNSTEADY
(i.e., TIME DEPENDENT)**

**THE APPROACH TO RESEARCH IS ALONG
TWO PARALLEL AND COMPLEMENTARY PATHS**



THE QUESTION FOR RESEARCH IS BALANCE

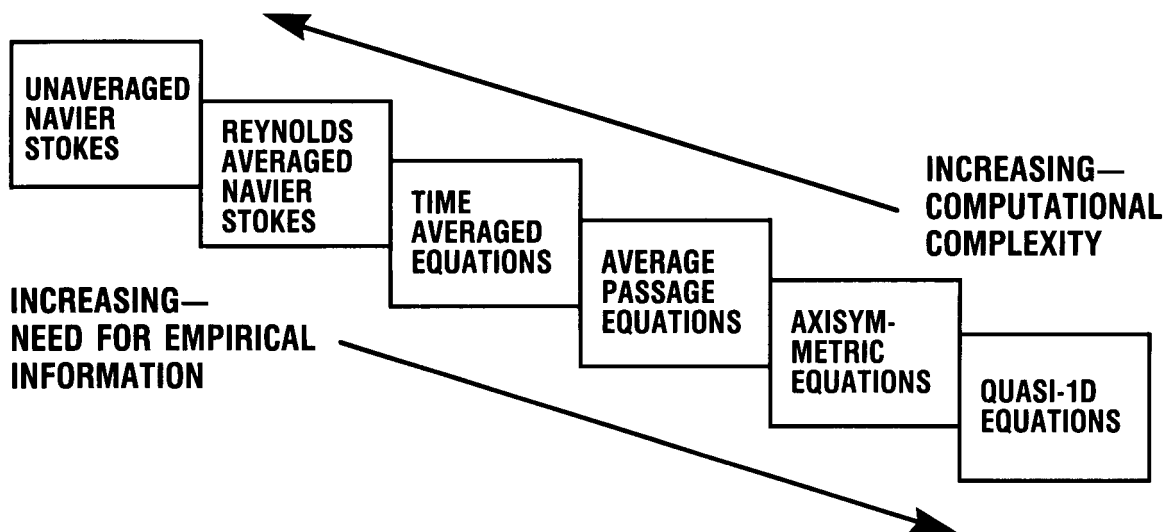
**WHEN TO WORK IN THE UNAVERAGED TIME
DOMAIN—AND WHEN AND HOW TO TIME AVERAGE?**

CD-87-28978

LEVELS OF COMPLEXITY ALONG THE PATH OF COMPUTATIONAL ANALYSIS

The time-accurate, unsteady Navier-Stokes and energy equations, which are capable of resolving all relevant time scales, describe the flow in a turbomachine. These are at the top of the stairs. They are the easiest set to formulate, but the most difficult set to solve because they require enormous computer power for the simplest cases. To ease the solution a variety of averages are taken. The critical step is to average properly by using the proper time and length scales. Each averaging step results in a loss of information or resolution in the equations, introduces more unknowns, and requires external input (obtained by experiment and by solving the exact equations for simpler cases). One can use engineering judgement to determine how much information is needed to complete the mathematical description of the particular problem at hand. The averaged equations allow this introduction of engineering judgement. The pure unaveraged Navier-Stokes equations must be solved in their entirety.

LEVELS OF COMPLEXITY ALONG THE PATH OF COMPUTATIONAL ANALYSIS

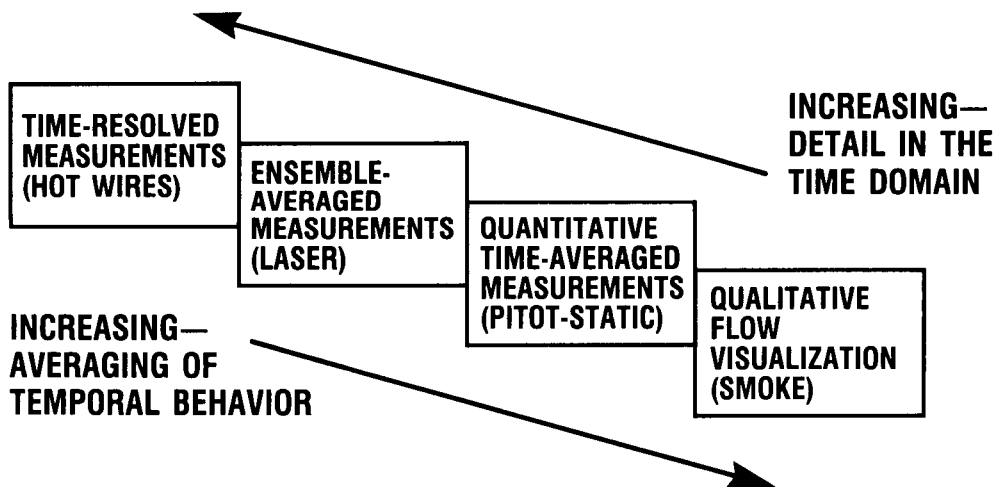


CD-87-28979

LEVELS OF COMPLEXITY ALONG THE PATH OF EXPERIMENTAL MEASUREMENTS

Both time-resolved and time-averaged measurements are essential to a full understanding of turbomachinery flow and heat-transfer characteristics. As with analysis, the problem is in determining which technique to apply and when to apply it. Frequently, an average result offers more insight than time-accurate detail. A qualitative visual observation may be crucial to understanding the essential physics. At other times, such a measurement may bury the essential physics. Traditionally, dynamic measurements have provided less accurate absolute measurements than average measurements. Recent improvements suggest that averaging time-resolved measurements may be more accurate than making average measurements, especially in heat transfer. Laser anemometry, in addition to offering the well-known nonintrusive advantage, has an especially nice feature of providing ensemble averages of the random statistics while retaining and identifying the deterministic time dependency. Ultimately, all levels are needed to do the job. The challenge is to choose the right tool for the right job.

LEVELS OF COMPLEXITY ALONG THE PATH OF EXPERIMENTAL MEASUREMENTS



AVERAGE PASSAGE FRAMEWORK OF EQUATIONS

The average passage equation system, developed by J. Adamczyk of NASA Lewis, is an example of averaging the Navier-Stokes equations. The three momentum equations and the energy equation are subjected to three averaging steps: one to remove random unsteadiness, a second to remove unsteadiness associated with blade-passing frequency and, finally, one to account for the uneven airfoil count from row to row. At its most fundamental level the averaging process introduces 11 unknowns in the axial momentum equation alone. The advantage is that the resulting equation set is much easier to solve mathematically. It will be necessary to conduct physical and/or numerical experiments to provide the correlations which bring closure to these equations. The properly averaged equations provide the framework for a large research effort into understanding the physics of fluid flow and heat transfer in turbomachines.

AVERAGE PASSAGE FRAMEWORK OF EQUATIONS

AXIAL MOMENTUM EQUATION:

$$\begin{aligned}
 & \frac{\partial}{\partial t_1} \lambda_1 \bar{\rho} \bar{r} \bar{V}_z + \frac{\partial}{\partial r} \lambda_1 \bar{r} \bar{\rho} \bar{V}_r \bar{V}_z + \frac{\partial}{\partial \theta} \lambda_1 \bar{\rho} \bar{V}_\theta \bar{V}_z + \frac{\partial}{\partial z} \lambda_1 \left(\bar{r} \bar{\rho} \bar{V}_z \bar{V}_z + \bar{r} \bar{P} \right) \\
 & \left. \begin{aligned}
 & = \frac{\partial}{\partial r} \lambda_1 \left(\bar{r} \bar{\rho} \bar{V}_z \bar{V}_z - r \bar{\rho} \bar{V}_r \bar{V}_z - r \bar{\rho} \bar{V}_z \bar{V}_z - r \bar{\rho} \bar{V}_z \bar{V}_z' \right) \\
 & + \frac{\partial}{\partial \theta} \lambda_1 \left(\bar{\rho} \bar{V}_z \bar{V}_z - \bar{\rho} \bar{V}_z \bar{V}_z - \bar{\rho} \bar{V}_z \bar{V}_z - \bar{\rho} \bar{V}_z \bar{V}_z' \right) \\
 & + \frac{\partial}{\partial z} \lambda_1 \left(\bar{r} \bar{\rho} \bar{V}_z \bar{V}_z - r \bar{\rho} \bar{V}_z \bar{V}_z - r \bar{\rho} \bar{V}_z \bar{V}_z - r \bar{\rho} \bar{V}_z \bar{V}_z' \right)
 \end{aligned} \right\} \\
 & + F_{IN}^{(ZR)} + F_V^{(ZR)} + F_{IN}^{(ZS)} + F_V^{(ZS)}
 \end{aligned}$$

CONVECTIVE TERMS

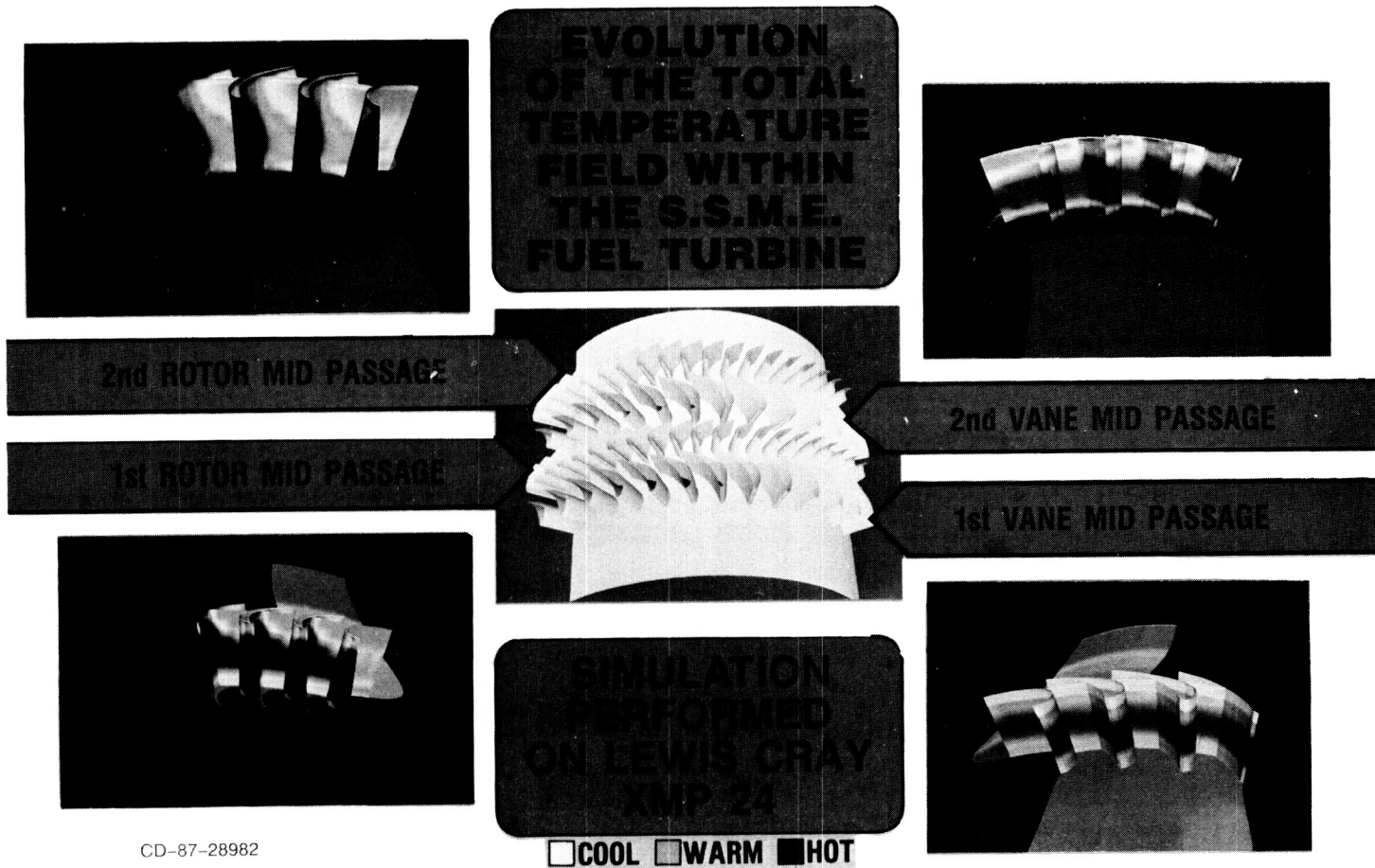
DIFFUSIVE TERMS

WHERE
 THE SUCCESSIVE OVERBARS
 REPRESENT AVERAGING OUT:
 (1) RANDOM UNSTEADINESS
 (2) PERIODIC UNSTEADINESS
 (3) UNEQUAL BLADE COUNT

**THE AVERAGING YIELDS A NEW EQUATION
 CONTAINING 11 UNKNOWNNS WHICH HAVE
 TO BE DETERMINED BY CONDUCTING
 PHYSICAL AND/OR NUMERICAL EXPERIMENTS**

**EVOLUTION OF THE TOTAL-TEMPERATURE FIELD WITHIN THE SPACE SHUTTLE MAIN
ENGINE (SSME) FUEL TURBINE**

An example of a calculation using the average passage system of equations depicts the evolution of a total-temperature distortion through the fuel pump turbine. The total-temperature profiles are color coded with red denoting regions of hot gas and blue denoting regions of cold gas. At this stage in the development of the average passage equation system many of the unknown viscous correlations remain to be determined and are set equal to zero for this calculation. The equations do, however, properly account for the interaction between blade rows. These results show that inviscid mixing as a result of streamwise vorticity generation can produce significant temperature differences between the suction and pressure side of a blade. This difference can lead to local regions of high thermal stress which can cause blade failure. The ability to capture the physics associated with this inviscid mixing process is a key element in increasing the durability of turbine blading.

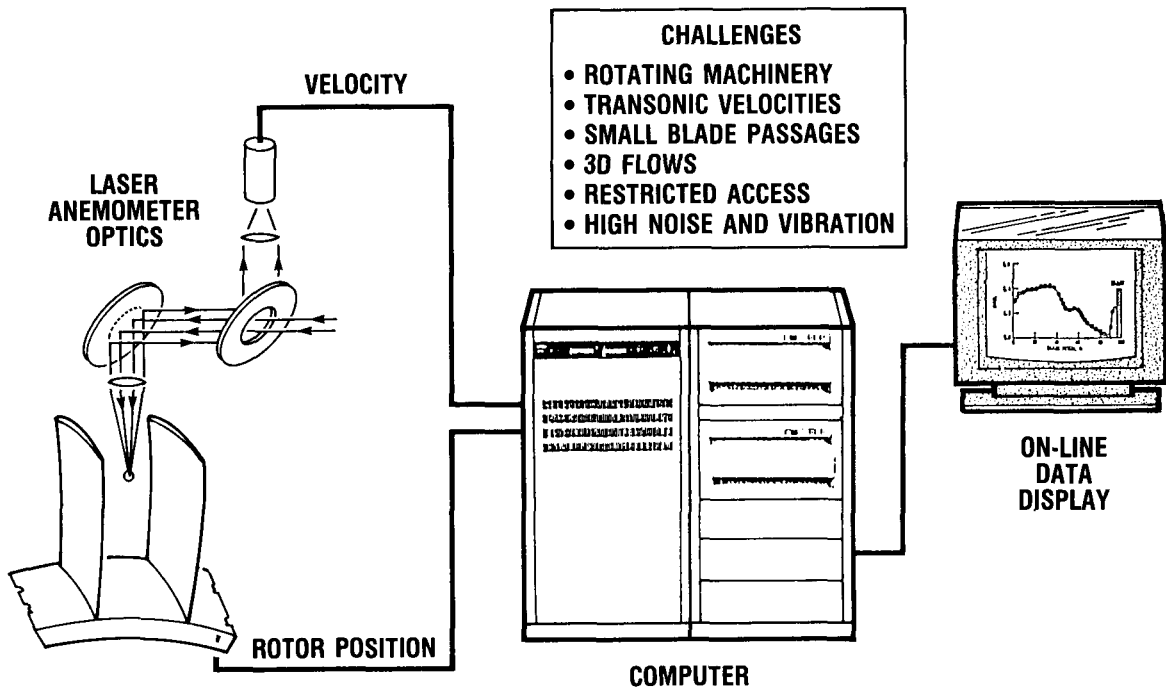


CD-87-28982

TURBOMACHINERY LASER ANEMOMETRY SYSTEMS

High-speed turbomachinery research facilities are characterized by high-speed rotating machines, small blade-passage heights, three-dimensional flows, transonic velocity levels, high noise and vibration levels, and restricted mechanical access. Because of its high spatial and temporal resolution and nonintrusive nature, laser anemometry has become the measurement method of choice for obtaining the detailed data required to assess the accuracy and sensitivity of flow analysis codes. In order to overcome the long measurement times required by laser anemometers and to capitalize on the detailed nature of the data which they generate, computer control of data acquisition and real-time data reduction and display are required. NASA Lewis is recognized as a world leader in the development and application of computer-controlled laser anemometer systems for use in both rotating and nonrotating turbine and compressor research applications.

TURBOMACHINERY LASER ANEMOMETRY SYSTEMS

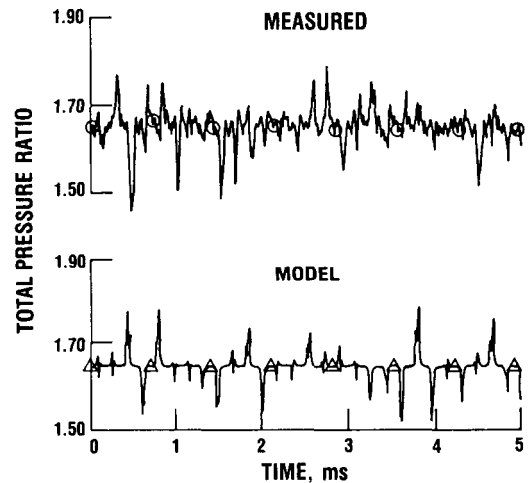
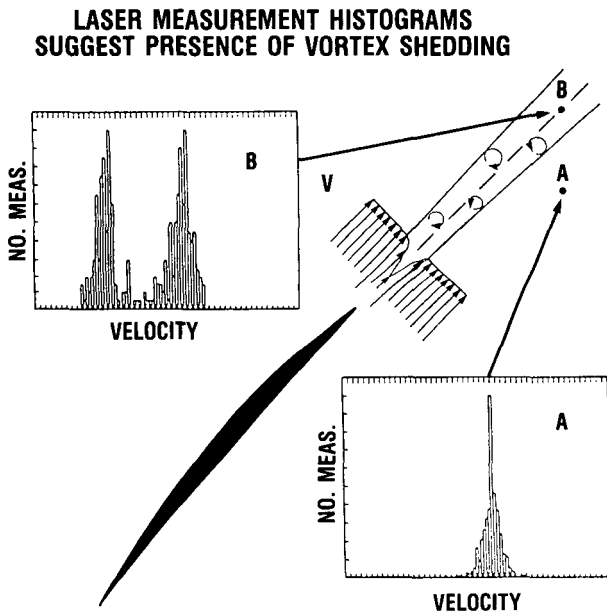


CD-87-28983

FAN ROTOR VORTEX SHEDDING - DEVELOPING MODELS TO EXPLAIN FLOW PHYSICS

Karman vortex streets are known to exist in blunt-body wakes over a wide flow regime. However, the existence of vortex streets in transonic fan and compressor-blade wakes was not generally anticipated since these blades have thin trailing edges. Laser anemometer measurements obtained in the wake of a transonic fan blade indicated two distinct states of the flow in the central portion of the blade wake - a high-velocity state and a low-velocity state. This behavior is consistent with that which would be displayed by a Karman vortex street. A simple vortex street model was constructed in an attempt to explain the experimental measurements. The model qualitatively agreed with the bimodal character of the velocity measurements. The model was also used to explain, for the first time, the highly unsteady nature of high-response pressure measurements made in the same wake flowfield. This research, which was a cooperative effort with MIT, typifies the manner in which advanced measurements coupled with simple modelling improve our understanding of complex flow phenomena.

FAN ROTOR VORTEX SHEDDING DEVELOPING MODELS TO EXPLAIN FLOW PHYSICS



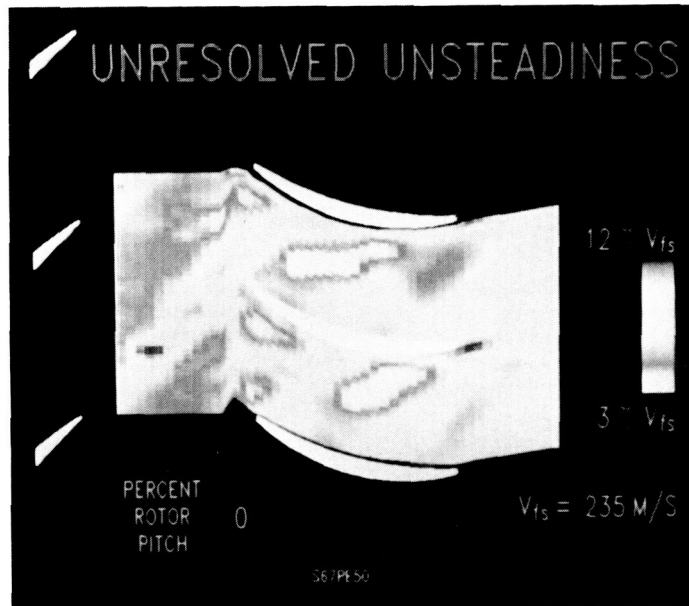
VORTEX WAKE MODEL DEDUCED FROM
THE LASER MEASUREMENTS AND
COMPARED TO HIGH-RESPONSE
PRESSURE PROBE DATA

COMPRESSOR "TURBULENT" KINETIC ENERGY DISTRIBUTION

Measurements of the unsteady flowfield within a compressor stator operating downstream of a transonic fan rotor have been obtained using laser anemometry. The figure shows a contour plot of the ensemble-averaged unresolved unsteadiness in the stator (which includes unsteadiness due to both turbulence and vortex shedding) for one rotor/stator relative position. Areas of high unresolved unsteadiness contain fluid which is in the rotor-blade wake. As the rotor blades rotate past the stator blades, the rotor wakes are convected through the stator row by the absolute flow velocity and, subsequently, chopped by the stator blades. Data obtained at additional times during the blade-passing cycle have been used to produce a movie sequence which illustrates the ensemble-averaged wake dynamics and its effect on the stator flowfield. We have the tools to measure temporal behavior in generic equivalents of a real machine.

COMPRESSOR "TURBULENT" KINETIC ENERGY DISTRIBUTION

ONE FRAME FROM DATA MOVIE SHOWING THE PROGRESS OF A WAKE THROUGH A STATOR PASSAGE DOWNSTREAM OF THE ROTOR



CD-87-28986

ORIGINAL PAGE IS
OF POOR QUALITY

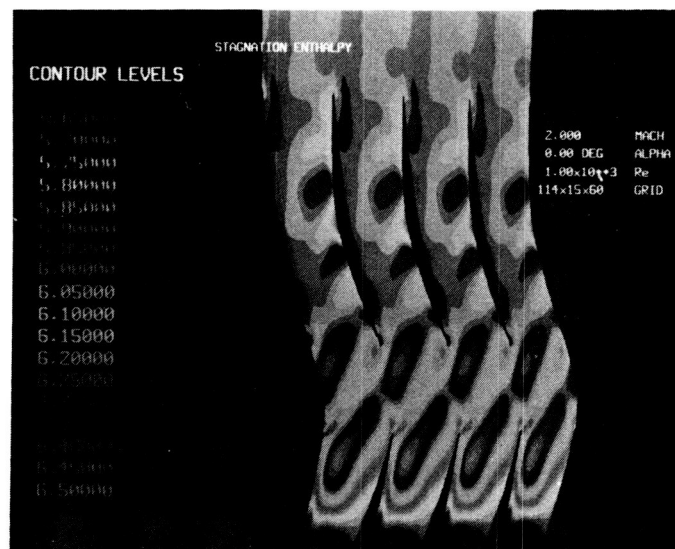
ORIGINAL PAGE IS
OF POOR QUALITY

UNSTEADY EULER CALCULATION IN A SUPERSONIC THROUGHFLOW FAN

A simulation was performed by David Whitfield and Mark Janius of Mississippi State University, in collaboration with NASA Lewis researchers, of the flow through a supersonic throughflow fan stage. This machine differs from today's machinery in that the axial Mach number is supersonic. For supersonic cruise, it offers an improvement in performance over transonic machinery. The three-dimensional simulation highlighted the rotor-stator interaction which occurs at design operating conditions. One of the interactions under study is the formation of "hot spots" at the leading edge of the stator. An animation illustrating the processes will be shown later in this conference. The "hot spots" are at a temperature which exceeds the instantaneous total temperature (absolute) of the flow stream exiting the rotor. Their formation is believed to be caused by the motion of the shock wave emanating from the pressure surface of the stator. We have the tools to compute temporal behavior in a real machine.

UNSTEADY EULER CALCULATION FOR A SUPERSONIC THROUGHFLOW FAN

ONE FRAME FROM A COMPUTATIONAL MOVIE SHOWING THE PROGRESS
OF THE SUPERSONIC ROTOR EXIT TOTAL TEMPERATURE
THROUGH THE DOWNSTREAM STATOR



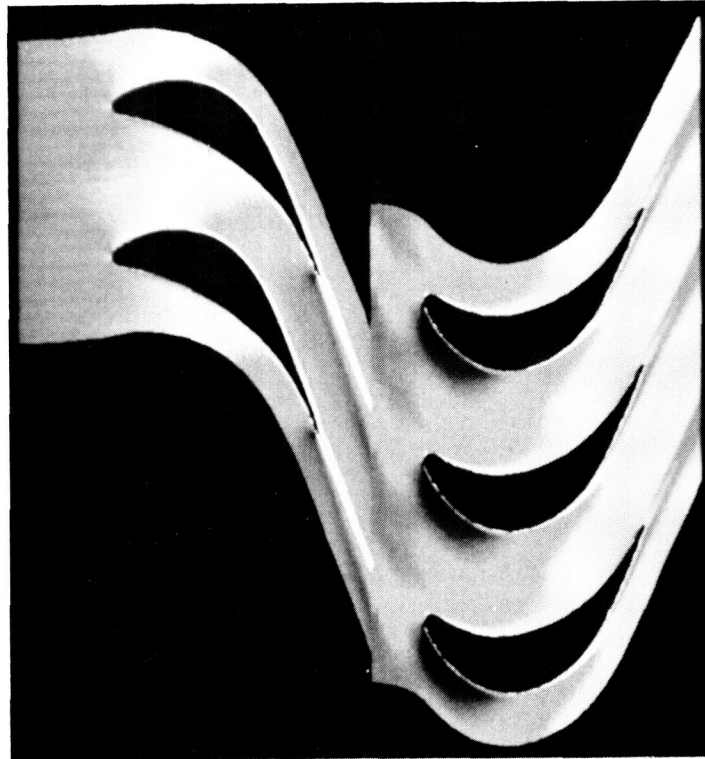
CD-87-28987

ROTOR/STATOR INTERACTION CALCULATION FOR THE SSME FUEL TURBINE

A NASA Lewis quasi-three-dimensional viscous code used to solve for the flow in an isolated turbomachinery blade row was modified to handle equal pitch stator/rotor interaction computations. The solution procedure has been applied to the first-stage turbine rotor of the space shuttle main engine (SSME) fuel turbopump. For this calculation, the upstream stator was scaled so that its pitch matched that of the rotor, and the pitch-to-chord ratio remained unchanged. A converged periodic solution was obtained after the stator had seen 10 passing rotor blades or 10 pitch rotations of the rotor, which takes about 2.5 hours on the Cray. Mach contours are shown in the figure for an equal pitch stator/rotor configuration. The average inlet Mach number to the upstream stator is 0.15. The wake region that develops behind the stator passes through the grid interface and is seen in the rotor computational domain. Currently, the analysis is being applied to multiple passages of a single-stage turbine. The stage airfoil configuration is two upstream stators followed by three rotors. The airfoil geometries are taken from the first stage of the SSME fuel turbopump and are scaled to 2:3 from their actual 41:63 airfoil ratio. The details of such interactions are important to our understanding of turbomachinery flows.

ROTOR/STATOR INTERACTION CALCULATION—SSME FUEL TURBINE

TIME ACCURATE 2D NAVIER-STOKES
CALCULATION, SHOWING THE MACH
CONTOUR DISTRIBUTION AT ONE
INSTANT OF TIME IN A BLADE-
PASSING PERIOD



CD-87-28988

EFFECT OF WAKES ON LAMINAR-TURBULENT TRANSITION IN A TURBINE STAGE

Detailed phase-resolved heat-flux data have been obtained on the blade of a full-stage rotating turbine (Teledyne 702) in work being done at CalSpan Corp. by Michael Dunn for Teledyne Corp. in conjunction with an Air Force-sponsored Vane-Blade Interaction program. A shock tube is used as a short-duration source of heated air, and platinum thin-film gages are used to obtain the heat-flux measurements. Some thin-film gages can be seen in the right-hand side of this figure in a leading-edge insert. Heat-flux results are presented in the left-hand side of the figure for the midspan at several locations along the suction surface from stagnation point to 78-percent wetted distance. Each phase-resolved plot represents the ensemble-average of about four to five vane-wake (or passage) crossings. The rapid decrease in heat flux level from stagnation point to trailing edge is evident, as well as the fluctuating laminar-to-turbulent (and back to laminar) component, as the rotor cuts the stator wakes. This high-frequency oscillation from laminar-to-turbulent flow has important implications for turbine heat-transfer analysis and can only be identified with sophisticated sensors, electronics, and dynamic signal analysis.

**EFFECT OF WAKES ON LAMINAR-TURBULENT TRANSITION
IN A TURBINE STAGE**

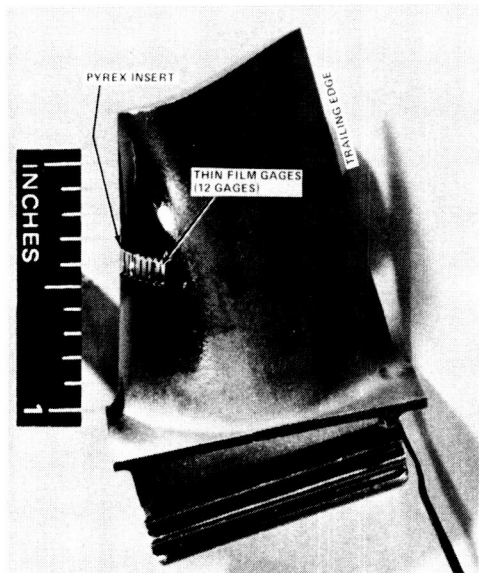
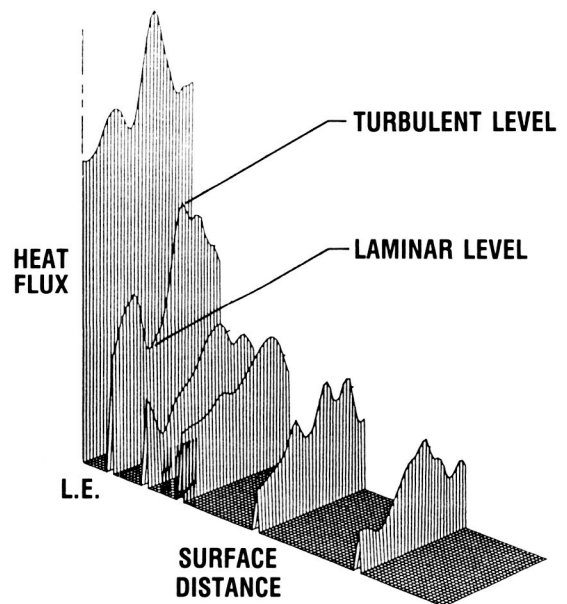


PHOTO OF A ROTOR BLADE INSTRUMENTED
WITH THIN-FILM SENSORS

CD-87-28989



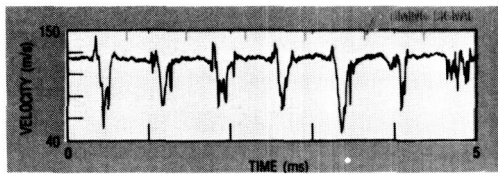
PHASE-RESOLVED CONTOUR PLOT
SHOWING THE PROGRESSION OF
TRANSITION ALONG THE AIRFOIL

UNSTEADY HEAT TRANSFER IN ROTOR-WAKE FLOWS

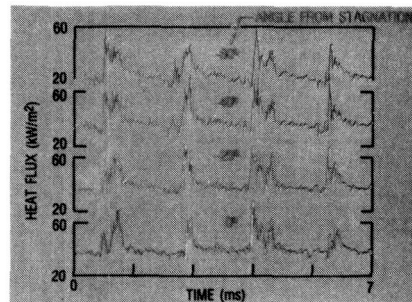
As a result of several recent studies, the nature and significance of wake-related flow unsteadiness in turbomachinery blading and its profound effect on heat transfer are beginning to be recognized. In particular, the enhancement of heat transfer to the stagnation region is of interest because of the critical importance of heat transfer in this region. The effect of freestream turbulence on time-average stagnation region heat transfer has been well documented. However, very few measurements have been obtained of the time-resolved effects of wake passage on heat transfer and the relationship of these effects to the corresponding velocity fluctuations. Current efforts are aimed at obtaining such measurements and performing statistical analysis to determine correlations between the unsteady velocity and heat-flux records. The figure shows a schematic of the rotor-wake rig and representative fluctuating velocity measurements. In addition, stagnation region unsteady heat-flux records are shown which reveal the high degree of heat transfer enhancement associated with each wake-passing event.

UNSTEADY HEAT TRANSFER IN ROTOR-WAKE FLOWS

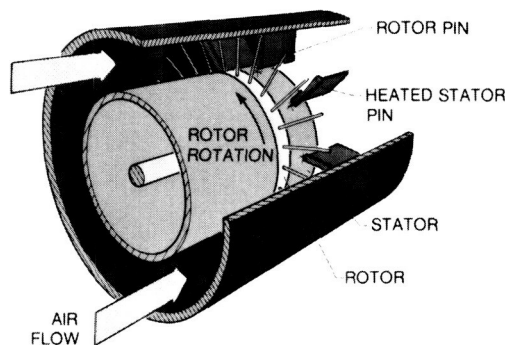
INSTANTANEOUS VELOCITY RECORD



SIMULTANEOUS HEAT FLUX RECORDS



ROTATING SPOKED
WHEEL PRODUCES
GOOD SIMULATION
OF AN AIRFOIL
TRAILING-EDGE
WAKE



INSTANTANEOUS
HEAT FLUX IN
STAGNATION
REGION IN WAKE
OF SPOKED ROTOR

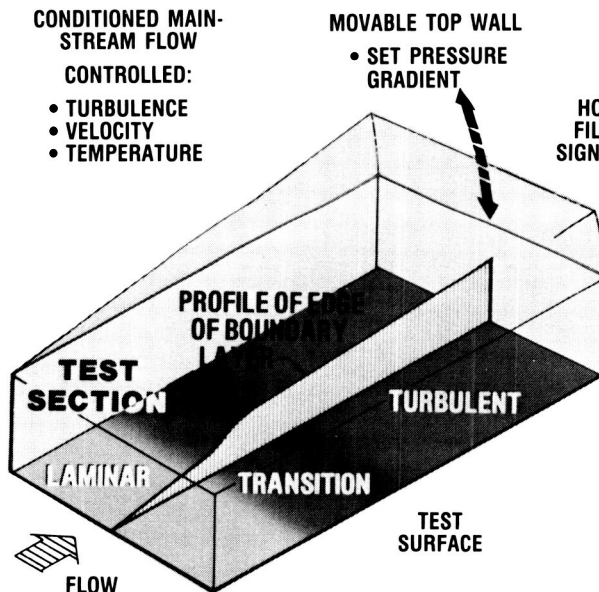
CD-87-28990

BOUNDARY-LAYER TRANSITION RESEARCH -- A STUDY OF INTERMITTENT BEHAVIOR

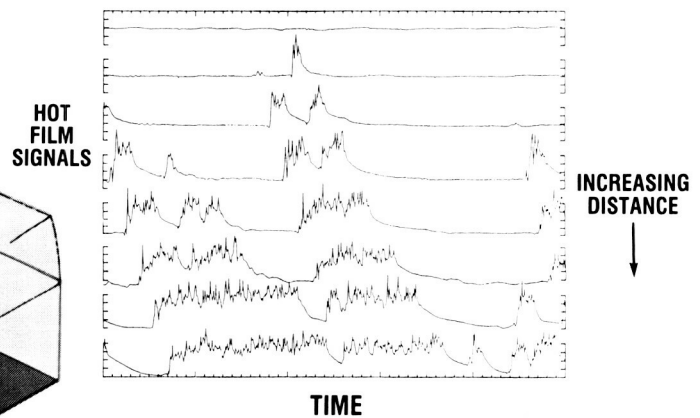
Since gas-side heat-transfer coefficients can vary by an order of magnitude within the transition region, a detailed understanding of boundary-layer transition is critical to the design of effective turbine-airfoil cooling schemes. This is particularly important when one realizes that such events can occur at blade-passing frequency. The present research program is aimed at understanding the fundamental differences between the "classical" boundary-layer transition process and the "bypass" transition process which occurs when a laminar boundary layer is perturbed by large freestream disturbances. The time-resolved hot wire velocity measurements obtained at a grid-generated freestream turbulence level of only 0.7 percent, indicate a bypass of small disturbances and the rapid development of a turbulent boundary layer as the flow progresses down the plate.

BOUNDARY-LAYER TRANSITION RESEARCH A STUDY OF INTERMITTENT BEHAVIOR

SKETCH OF TEST SECTION SHOWING MAJOR FEATURES OF FACILITY



FREE-STREAM TURBULENCE LEVEL 0.77%



CARPET PLOT OF SIMULTANEOUS HOT FILM RECORDS, SHOWING PROGRESSION AND GROWTH OF TURBULENCE ALONG THE PLATE SURFACE

CD-87-28991

HIGH-RESOLUTION, LIQUID-CRYSTAL, TURBINE-ENDWALL HEAT-TRANSFER DATA

The photograph of the experimental color temperature patterns shows contours of constant temperature obtained from an experimental technique in which a uniformly heated turbine-vane cascade endwall surface is operated in an air flow. Resulting isotherms on the test surface (which are also lines of constant heat-transfer coefficient) are indicated by thermochromic liquid crystals. The photographic data was then digitized for computer-based processing and display, to show color contours of Stanton Number (nondimensional heat-transfer coefficient). The highest heat transfer rates occur in the vane stagnation region (shown in red). This computer-generated display can be used for direct comparison with code-generated predictions. Complex phenomena, such as these, require complex analyses - the three-dimensional Navier-Stokes codes.

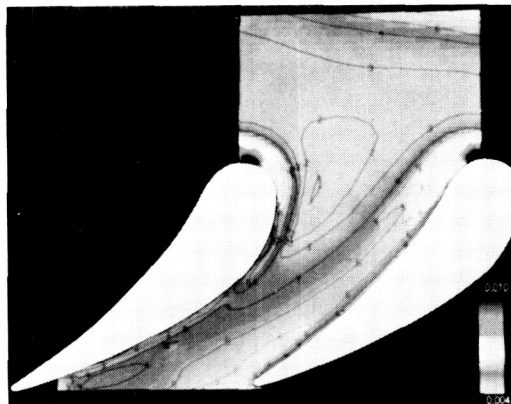
HIGH-RESOLUTION LIQUID-CRYSTAL TURBINE ENDWALL HEAT TRANSFER DATA

COMPUTER-BASED COLOR CONTOURS OF THE
ENDWALL HEAT TRANSFER DISTRIBUTION

EXPERIMENTAL LIQUID-
CRYSTAL COLOR
TEMPERATURE PATTERNS



(ISOTHERMS
AT CONSTANT
HEAT FLUX)



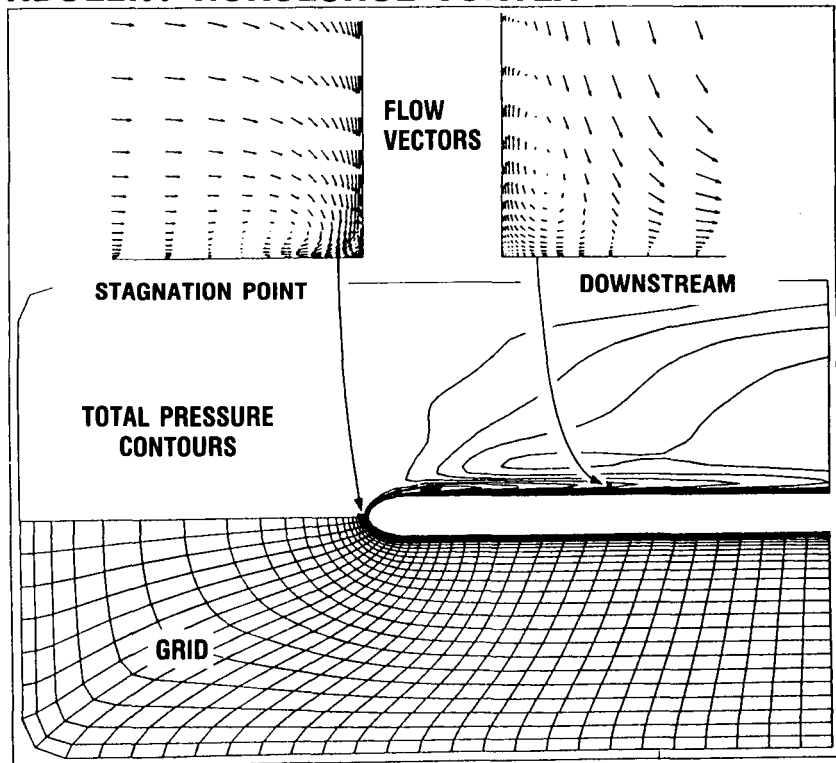
(DIGITIZED COMPOSITE
OF EXPERIMENTAL DATA)

THREE-DIMENSIONAL NAVIER-STOKES CODE ANALYSES OF THE TURBULENT HORSESHOE VORTEX

A three-dimensional Navier-Stokes analysis code (RVC3D) is being developed for turbomachinery blade rows. The Navier-Stokes equations are written in a body-fitted coordinate system rotating about the x-axis. Streamwise viscous terms are neglected by using the thin-layer assumption, and turbulence effects are modeled with the Baldwin-Lomax eddy viscosity. The equations are discretized by using finite differences, and solved by using a multistage Runge-Kutta algorithm with a spatially varying time step and implicit residual smoothing. Calculations have been made of a horseshoe vortex formed at the junction of a turbulent endwall boundary layer and a blunt fin. This geometry is to be tested experimentally later this year by members of the Heat Transfer Branch. The calculations were done on a 65 by 33 by 25 grid (lower figure) at the nominal tunnel operating conditions (Mach number = 0.6, fin thickness Reynolds number = 260 000, inlet boundary-layer thickness = 1 in). Total pressure contours, 0.025 in. above the endwall, show the primary vortex core (center figure). Vector plots on the upstream symmetry plane (upper left) and on a downstream cross-channel plane (upper right) show the development of a double vortex system. The calculations required about one million words of storage and 10 min of CPU time on a Cray X-MP.

3D NAVIER-STOKES CODE ANALYSIS OF THE TURBULENT HORSESHOE VORTEX

**CALCULATION OF
HORSESHOE VORTEX
FOR SAME GEOMETRY
AND CONDITIONS OF
THE COMPRESSIBLE
FLOW TUNNEL**

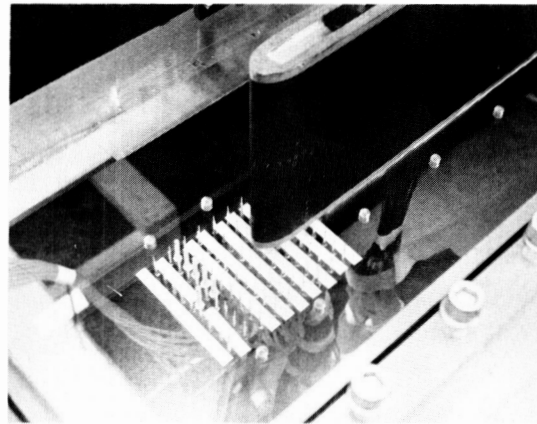
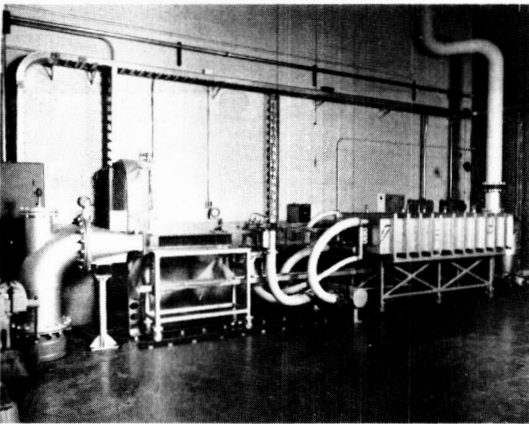


CD-87-28994

THREE-DIMENSIONAL COMPRESSIBLE FLOW TUNNEL

This new facility for fluid mechanics and heat-transfer research will provide benchmark quality experimental data for internal flow code validation. Focus will be on the three-dimensional interaction of the intersecting model with the surface plate (i.e., the horseshoe vortex). The facility will have the following capabilities: (1) maximum Mach number of 0.6; (2) no tunnel side-wall boundary layers; (3) controllable boundary layer on top and bottom walls of the tunnel; and (4) low inlet turbulence (less than 0.5 percent). The first phase of the experiment will consist of three parts: (1) fluid mechanics measurements will be taken by using a five-hole probe and hot film shear-stress gauges, (2) various flow visualization techniques will be used to define the flow path at the intersecting surfaces, and (3) heat transfer data will be recorded by the liquid crystal technique. The second phase of the work, to be conducted in about two years, will include full flowfield measurements by laser anemometry. This experiment and the RVC3D code development form a critical partnership to the successful development of a turbomachinery analysis capability.

3D COMPRESSIBLE FLOW TUNNEL



OBJECTIVE

TO OBTAIN DATA TO VERIFY COMPUTATIONAL FLUID MECHANICS COMPUTER CODES THAT ARE CAPABLE OF SOLVING FULLY 3D FLOWS INCLUDING HEAT TRANSFER

LARGE LOW-SPEED CENTRIFUGAL COMPRESSOR, NEW FLOW-PHYSICS AND CODE-VALIDATION RIG

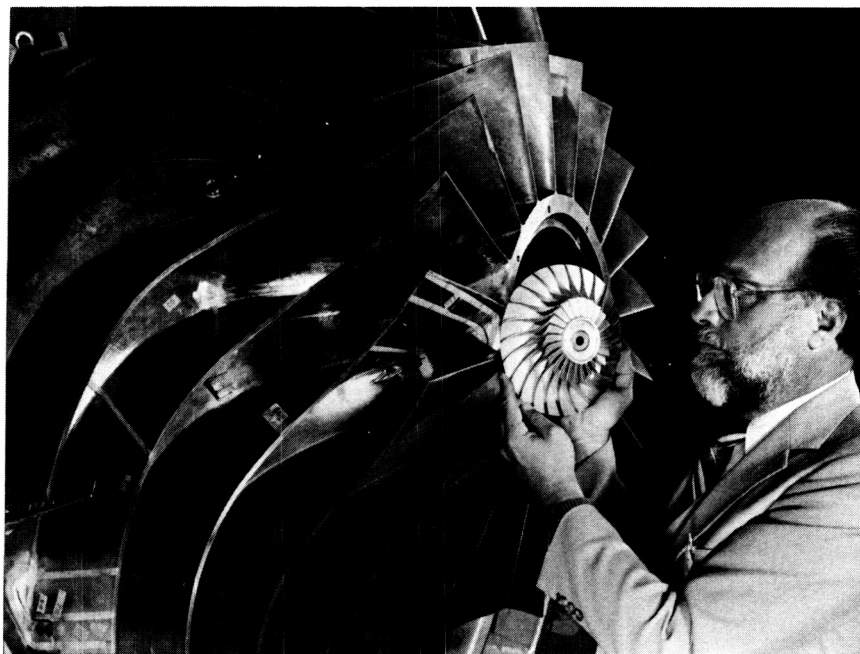
Centrifugal compressors feature large surface area and small exit-passage heights. Viscous flow effects, therefore, have a major impact on the flowfield within centrifugal compressors. The inability to accurately predict and measure these flows contributes in large part to the inherently lower efficiency of centrifugal compressors relative to axial-flow compressors.

The large low-speed centrifugal compressor shown in the background of the photograph has been designed specifically to provide flow modelling and viscous code-validation data for centrifugal compressors. The impeller was designed to be aerodynamically similar to high-performance, high-speed centrifugal compressors such as the small 6:1 pressure-ratio impeller shown in the photograph. The low-speed impeller has a tip diameter of 50 inches and a rotational speed of 1950 rpm. Inlet and exit blade heights are 9 inches and 4.75 inches, respectively. The large size and low speed of the new impeller generate viscous flow regions (such as blade and endwall boundary layers) and tip clearance flows which are large enough to measure in detail with laser anemometry.

LARGE LOW-SPEED CENTRIFUGAL COMPRESSOR NEW FLOW PHYSICS AND CODE VALIDATION RIG

**UNDERSTANDING THE FLOW
PHYSICS AND THE VALIDATION
OF COMPLEX TURBOMACHINERY
3D NAVIER-STOKES CODES
REQUIRES VERY LARGE ROTA-
TING MACHINERY IN ORDER TO
BE ABLE TO EXAMINE THE
VISCIOUS BOUNDARY LAYERS**

**SIZE COMPARISON BETWEEN
LOW-SPEED IMPELLER AND
2-LB/S HIGH-SPEED IMPELLER**

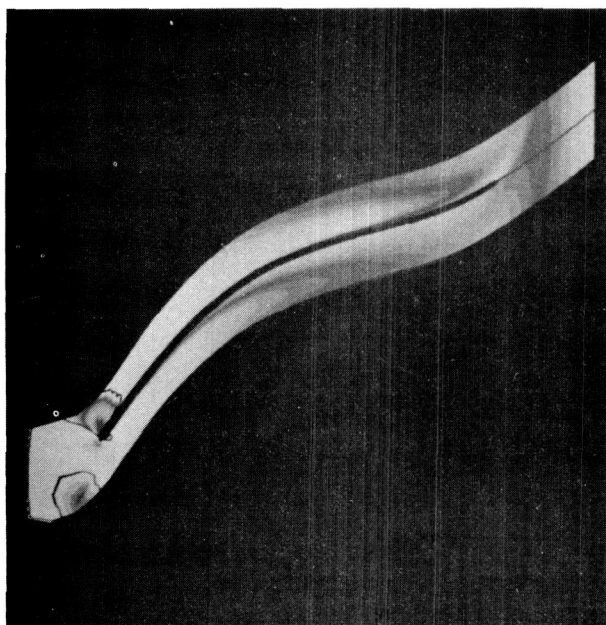


CD-87-28996

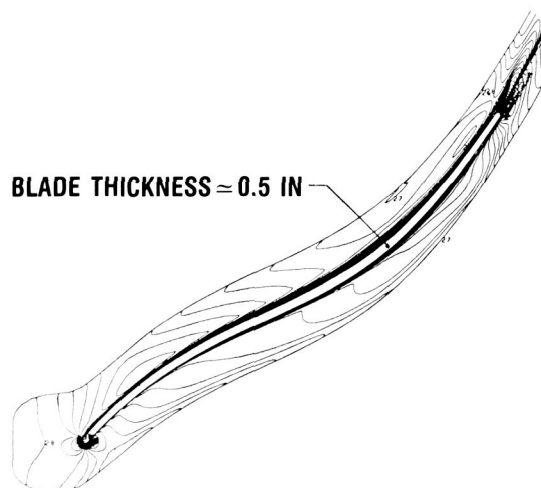
QUASI-THREE-DIMENSIONAL NAVIER-STOKES CODE FOR TURBOMACHINERY ANALYSIS

An efficient Navier-Stokes analysis code (RVCQ3D) has been developed for turbomachinery. The effects of radius change, stream surface thickness, and rotation are included, which allows calculations of centrifugal impellers, radial diffusers, and axial machines with contoured endwalls. The unsteady Navier-Stokes equations are solved in finite-difference form using an explicit Runge-Kutta algorithm with a spatially varying time step and multigrid convergence acceleration. The flow in a 6:1 pressure-ratio centrifugal impeller has been calculated on a 161 by 33 grid. Relative Mach number contours in the figure show a supersonic bubble on the leading edge terminated by a shock. Rotational effects make the suction-surface boundary layer thin, the pressure-surface boundary layer thick, and they cause the wake to leave the trailing edge in a spiral. The calculations required about 3 000 000 words of storage and 1.5 min of CPU time on a Cray X-MP. The flow in the new, large low-speed rig has also been calculated and is shown on the right. The ability to validate this and other codes in the large low-speed rig will generate confidence in the high-speed calculations where validation is near to impossible.

QUASI-3D NAVIER-STOKES CODE FOR TURBOMACHINERY ANALYSIS



**ANALYSIS OF HIGH-SPEED 6:1
CENTRIFUGAL IMPELLER**



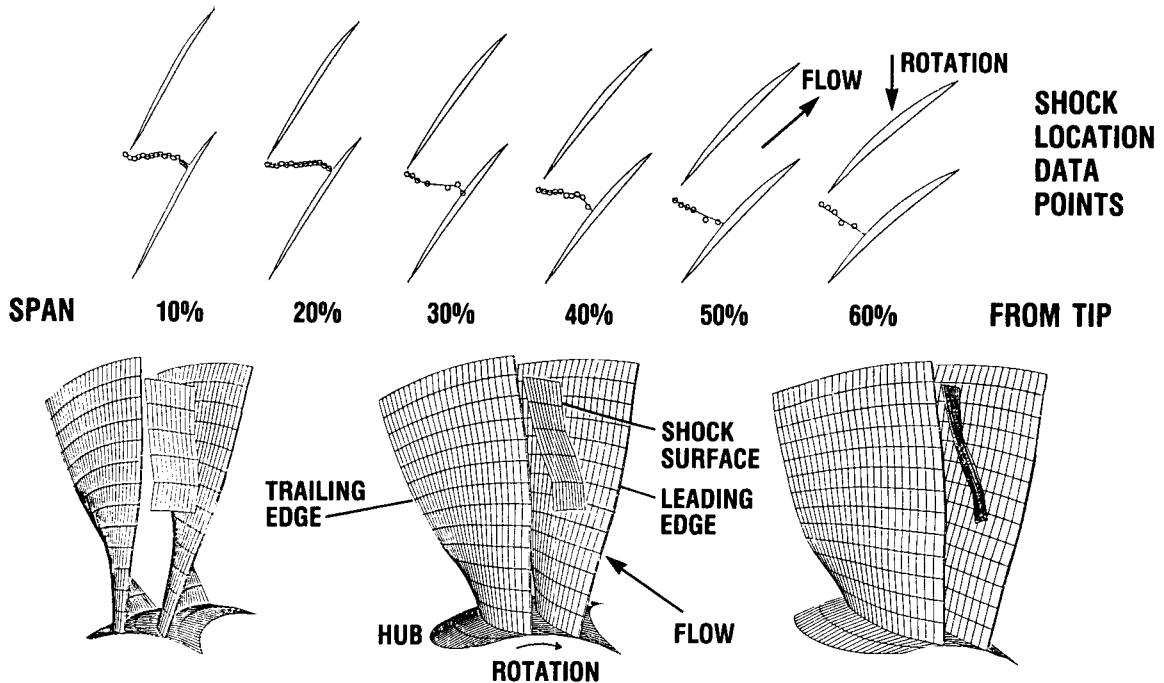
**ANALYSIS OF LARGE LOW-SPEED
CENTRIFUGAL IMPELLER**

CD-87-28997

MEASUREMENT OF THREE-DIMENSIONAL SHOCK STRUCTURE IN A TRANSONIC AXIAL-FLOW FAN

In order to fully validate the codes, they must be shown capable of capturing real physics in real machines. An example is the shock behavior in a transonic fan. Many axial fan and compressor design systems currently in use do not account for passage shock three-dimensionality. In addition, preliminary blade designs are often performed as two-dimensional calculations on blade-to-blade stream surfaces. An assesement of the shock three-dimensionality in transonic rotors is therefore necessary in order to properly account for three-dimensional effects in the design process. Shock locations determined from laser anemometer measurements are shown on blade-to-blade surfaces of revolution in the upper half of the figure. Three different views of the same data, as displayed on a graphics workstation, are shown in the lower half of the figure. A significant spanwise lean of the shock surface is evident in these three-dimensional views.

MEASUREMENT OF 3D SHOCK STRUCTURE IN A TRANSONIC AXIAL-FLOW FAN



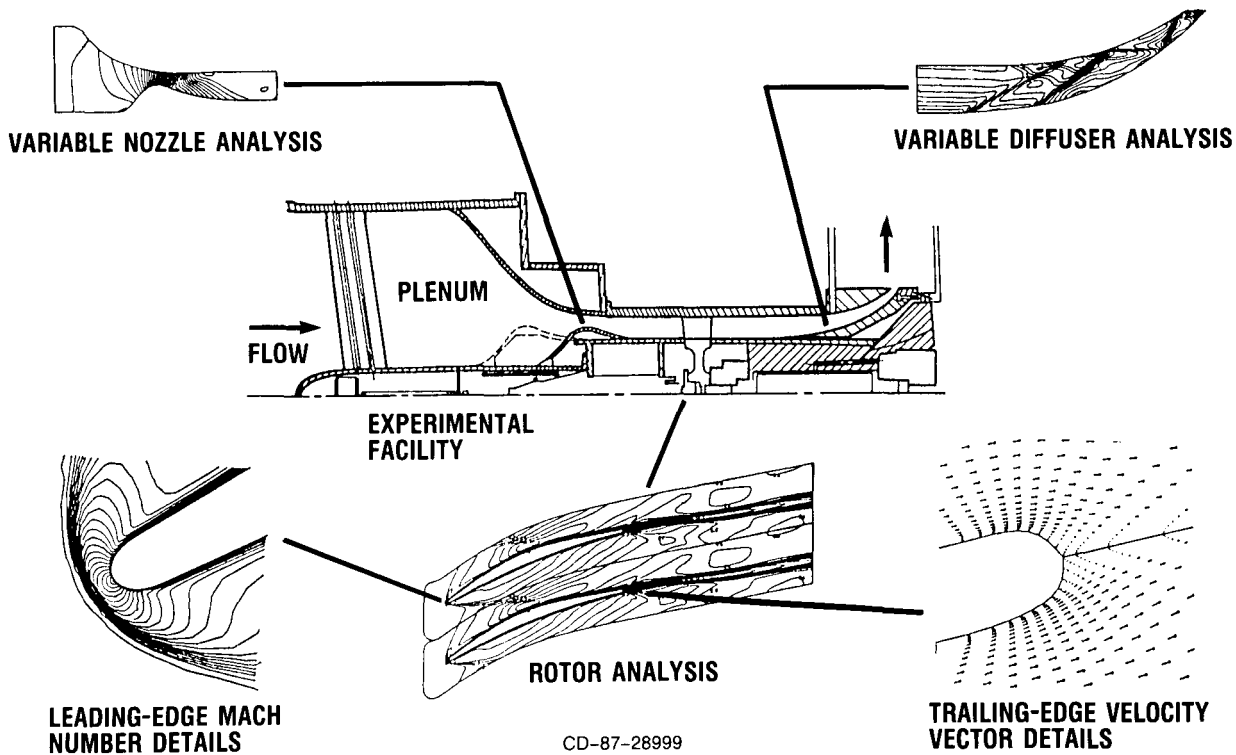
DATA ENHANCEMENT AND 3D VIEWING ROTATION

CD-87-28998

APPLICATION OF ADVANCED CODES FOR DESIGN OF A SUPERSONIC THROUGHFLOW FAN EXPERIMENT

Increased emphasis on sustained supersonic cruise or hypersonic cruise has revived interest in the supersonic throughflow fan as a possible component in advanced propulsion systems. Use of a fan that can operate with a supersonic inlet axial Mach number is attractive from the standpoint of reducing the inlet losses incurred in diffusing the flow from a supersonic flight Mach number to a subsonic one at the fan face. The data base for components of this type is practically nonexistent, and design of any experiment to study feasibility of this concept must rely heavily upon advanced computational tools to enhance the possibility of success. Computer codes that have been developed for design and analysis of transonic turbomachines were modified to allow calculations of blade rows with supersonic inlet Mach numbers. An inviscid/viscous code and a parabolized viscous code were used to design and analyze the variable nozzle and variable diffuser necessary for the experiment. Off-design analysis of the various components of the experiment indicated that all components would operate as expected over the flow and speed range of the experiment. The figure shows the results obtained for the inlet variable nozzle which sets up the inlet flowfield, the fan rotor, and the variable diffuser which decelerates the flow toward the collector inlet. All components were analyzed with two different codes in order to give increased confidence in the computed results. This is the ultimate goal of the turbomachinery program - to develop the tools which allow us to push beyond our experience with confidence.

APPLICATION OF ADVANCED CODES FOR DESIGN OF SUPERSONIC THROUGHFLOW FAN EXPERIMENT



POSITION OF NASA LeRC TURBOMACHINERY PROGRAM ON THE COMPUTATIONAL
AND EXPERIMENTAL PATHS

A complete and mature program has research at all levels along the computational and experimental paths. In the NASA LeRC turbomachinery program, special emphasis is placed on the analytic range from the ensemble (Reynolds) averaged Navier-Stokes equations to the average passage equations. The experimental emphasis is on high-response time-resolved measurements and in real machinery laser anemometry measurements. It is important to emphasize that the successful application of these tools will require a strong interaction between the computational and experimental paths.

**POSITION OF THE TURBOMACHINERY PROGRAM ON
COMPUTATIONAL AND EXPERIMENTAL PATHS**

