Aeropropulsion '87
Session 3—
Internal Fluid Mechanics Research

Preprint for a conference at
NASA Lewis Research Center
Cleveland, Ohio, November 17–19, 1987
Aeropropulsion '87
Session 3—Internal Fluid Mechanics Research

Preprint for a conference at
NASA Lewis Research Center
Cleveland, Ohio, November 17-19, 1987
## CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Authors</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>INTERNAL FLUID MECHANICS</td>
<td>Brent A. Miller and Louis A. Povinelli</td>
<td>3-1</td>
</tr>
<tr>
<td>INLETS, DUCTS, AND NOZZLES</td>
<td>John M. Abbott, Bernhard H. Anderson, and Edward J. Rice</td>
<td>3-52</td>
</tr>
<tr>
<td>TURBOMACHINERY</td>
<td>Robert J. Simoneau, Anthony J. Strazisar, Peter M. Sockol, Lonnie Reid, John J. Adamczyk</td>
<td>3-29</td>
</tr>
<tr>
<td>CHEMICAL REACTING FLOWS</td>
<td>Edward J. Mularz and Peter M. Sockol</td>
<td>3-53</td>
</tr>
</tbody>
</table>

PRECEDING PAGE BLANK NOT FILMED
INTERNAL FLUID MECHANICS

Brent A. Miller
and
Louis A. Povinelli

ABSTRACT

Internal Fluid Mechanics research at Lewis is directed toward an improved understanding of the important flow physics affecting aerospace propulsion systems, and applying this improved understanding to formulate accurate predictive codes. To this end, research is conducted involving detailed experimentation and analysis. The presentations to follow summarize ongoing work and indicate future emphasis in three major Research Thrusts, namely, Inlets, Ducts and Nozzles; Turbomachinery; and Chemical Reacting Flows.
GOAL

Achievement of this goal requires that carefully planned and executed experiments be conducted in order to develop and validate useful codes. It is also important that the ultimate user community be part of the process in order to assure relevancy of the work and to hasten its practical application.

GOAL

TO BRING INTERNAL COMPUTATIONAL FLUID MECHANICS TO A STATE OF PRACTICAL APPLICATION FOR AEROSPACE PROPULSION SYSTEMS
It is critical that numerical code development work and experimental work be closely coupled. The insights gained are represented by mathematical models that form the basis for code development. The resultant codes are then tested by comparing them with appropriate experiments in order to ensure their validity and determine their applicable range.
RESEARCH THRUSTS

Propulsion systems are characterized by highly complex and dynamic internal flows. Many complex, three-dimensional flow phenomena may be present, including unsteadiness, shocks, and chemical reactions. By focusing on specific portions of a propulsion system, it is often possible to identify the dominant phenomena that must be understood and modeled for obtaining accurate predictive capability. In the following presentations on Internal Fluid Mechanics, research emphasis is placed on Inlets, Ducts and Nozzles, on Turbomachinery, and on Chemical Reacting Flows. These three Research Thrusts serve as a focus leading to greater understanding of the relevant physics and to an improvement in analytical tools. This in turn will hasten continued advancements in propulsion system performance and capability.
INLETS, DUCTS, AND NOZZLES

John M. Abbott,
Bernhard H. Anderson,
and
Edward J. Rice

ABSTRACT

The internal fluid mechanics research program in inlets, ducts, and nozzles is described. The program consists of a balanced effort between the development of computational tools (both parabolized Navier-Stokes and full Navier-Stokes) and the conduct of experimental research. The experiments are designed to better understand the fluid flow physics, to develop new or improved flow models, and to provide benchmark quality data sets for validation of the computational methods.

The inlet, duct, and nozzle research program is described according to three major classifications of flow phenomena: (1) highly three-dimensional flow fields, (2) shock – boundary-layer interactions, and (3) shear layer control. Specific examples of current and future elements of the research program are described for each of these phenomena. In particular, the highly three-dimensional flow field phenomena is highlighted by describing the computational and experimental research program in transition ducts having a round-to-rectangular area variation. In the case of shock – boundary-layer interactions, the specific details of research for normal shock – boundary-layer interactions are described. For shear layer control research in vortex generators and the use of aerodynamic excitation for enhancement of the jet mixing process are described.

Future research in inlets, ducts, and nozzles will include more emphasis on three-dimensional full Navier-Stokes methods and corresponding experiments designed to concentrate on the appropriate three-dimensional fluid flow physics.
The internal fluid mechanics research program in inlets, ducts, and nozzles is described according to three types of fluid flow phenomena—highly three-dimensional flow fields, shock-boundary-layer interactions, and shear layer control. The importance of each of these flow phenomena is a result of the drivers listed on the figure. For example, highly three-dimensional internal flow fields result from unconventional engine locations where twisting and turning inlets, ducts, and nozzles must be designed to deliver the airflow to and from the free stream. Aircraft thrust vectoring requirements quite often lead to the transitioning of nozzle cross-sectional geometries from round to rectangular with resultant three-dimensional flows. Aircraft maneuverability requirements can lead to significant three-dimensional flow fields entering the propulsion system inlet and ducting system. The push toward higher flight speeds, for both military and civilian applications, leads to the importance of research in shock-boundary-layer interactions within inlets, ducts, and nozzles. The desire to design inlets, ducts, and nozzles to be as short and light as possible points to the importance of shear layer control as a means for "stretching" the limits of the geometry while avoiding internal flow separations. Shear layer control in another sense, that is, the use of aerodynamic excitation to control the formation and development of a mixing layer, offers the potential for enhancing the mixing process in external nozzle flows.
Specific elements of the inlet, duct, and nozzle research program are listed for each of the three flow field phenomena. The four elements highlighted in the figure will be expanded upon in the remainder of the presentation. Specifically, highly three-dimensional flow fields will be illustrated by describing the transition duct research program. The shock-boundary-layer interaction phenomena will be illustrated with a description of the normal shock-boundary-layer interaction research program. Shear layer control research will be illustrated with two examples of current programs: vortex generator research and enhanced jet mixing research. The remaining four elements of the overall program, that is, offset ducts, oblique shock-boundary-layer interactions, glancing sidewall shock-boundary-layer interactions, and boundary-layer bleed, will not be described in this presentation, although they are also key elements of the overall program.

INLETS, DUCTS, AND NOZZLES
FLOW PHENOMENA

- HIGHLY 3D FLOW FIELDS
  - TRANSITION DUCTS
  - OFFSET DUCTS

- SHOCK-BOUNDARY-LAYER INTERACTIONS
  - NORMAL SHOCK-BOUNDARY LAYER
  - OBLIQUE SHOCK-BOUNDARY LAYER
  - GLANCING SIDEWALL SHOCK-BOUNDARY LAYER

- SHEAR LAYER CONTROL
  - VORTEX GENERATORS
  - BOUNDARY-LAYER BLEED
  - ENHANCED JET MIXING
The approach to the research program in inlets, ducts, and nozzles consists of a balance between computational and experimental research. Computationally, much of the emphasis until recently has been on the development and validation of parabolized Navier-Stokes methods. More recently and for the future, more emphasis is being placed on three-dimensional full Navier-Stokes methods.

Shown on the right-hand side of the figure are the elements of the experimental program in inlets, ducts, and nozzles. They are aligned with the respective computational method and point out the close linkage between the computational and experimental elements of the program. Note that each of the program elements listed in the previous figure under the three different flow phenomena also appear here as specific experiments.
TRANSITION DUCT RESEARCH

Highly three-dimensional flow field research will be illustrated by describing the transition duct research program.

INLETS, DUCTS, AND NOZZLES
FLOW PHENOMENA

• HIGHLY 3D FLOW FIELDS
  - TRANSITION DUCTS
  - OFFSET DUCTS

• SHOCK-BOUNDARY-LAYER INTERACTIONS
  - NORMAL SHOCK-BOUNDARY LAYER
  - OBLIQUE SHOCK-BOUNDARY LAYER
  - GLANCING SIDEWALL SHOCK-BOUNDARY LAYER

• SHEAR LAYER CONTROL
  - VORTEX GENERATORS
  - BOUNDARY-LAYER BLEED
  - ENHANCED JET MIXING
A sample three-dimensional parabolized Navier-Stokes (PNS) computation for a specific transition duct geometry is illustrated in this figure. The geometry is described by a rectangular exit having an aspect ratio (width/height) of 3.0. The duct is 3 inlet diameters long and has an exit-to-inlet area ratio of 1.0. The figure shows near-wall velocity vectors on the left with a display of the secondary velocity vectors in the rectangular exit plane shown below it. The three-dimensional character of the flow field is clear. On the right side of the figure are contours of surface shear stress on the duct. Zero shear stress corresponds to the onset of flow separation on the duct surface. Note that the computation indicates that this particular geometry has a small, localized separation zone about halfway down the duct. The flow reattaches just downstream of this zone and remains attached for the remainder of the duct length. The separation is apparent in both the near-wall velocity vectors and in the surface shear stress contours.
As an illustration of how the computation results shown in the previous figure can be used, this figure shows the internal flow separation bounds for a class of transition duct geometries. Each duct geometry has the same area ratio (1.0) but the length ratio and the exit aspect ratio are varied. Specifically, for four different aspect ratios, the length ratio was decreased from a value where the internal flow was completely attached to a value where the flow just began to separate. This series of computations then resulted in the separation bound shown in the figure. For geometries above the curve, the flow is attached, for geometries below the curve, the flow is separated. One experiment data point is spotted on the figure as a case where the flow was massively separated within the duct.
To develop a more detailed understanding of the flow physics within transition ducts, to improve models of the flow physics, and to validate computations like those shown in the previous two figures, experiments are underway using the model and facility shown in this figure. The model was machined to match exactly one of the geometries for which the three-dimensional PNS method had been applied. Shown in this figure is a photograph of the model during the final machining stage along with a superimposed display of the computational surface. Also shown is the test facility. Special care was taken to condition the transition duct inflow properly to provide the desired levels of inflow turbulence, flow angularity, and boundary-layer profile. Modifications to the facility are currently underway whereby the inflow tank is being connected to a high-pressure supply system to provide higher Reynolds number test capability.
Experimental results are shown in this figure at a one-dimensional duct Mach number of 0.5. On the left side of the figure, surface static-pressure measurements are shown along the centerline of the duct. On the right side of the figure are shown results from a surface-oil-streak flow-visualization experiment. These oil streaks agree quite well qualitatively with the computational near wall velocity vectors. Although not shown here, experimental measurements have just been made of flow direction in the duct exit plane. Efforts are currently underway to compare those results with the corresponding three-dimensional PNS computations. Future experiments will include detailed probing of the flow field within this duct and others and direct measurement of surface shear stress using an advanced laser measurement technique.
In addition to conducting aerodynamic experiments with the highly three-dimensional flow fields of transition ducts, heat-transfer experiments are also being conducted. Heat transfer is of particular concern for these types of flow fields because of applications where the three-dimensional flows may result in high-temperature flow streams (i.e., engine core flow) finding their way to the transition duct (nozzle) surfaces. The figure shows a square-to-rectangular transition duct model which was tested in the facility shown on the right. To measure the surface heat-transfer characteristics of the duct, the surface was coated with a liquid-crystal material. After establishing the desired amount of airflow through the duct, the duct surface was heated so that the resulting color bands (isotherms) on the liquid-crystal surface could be interpreted in terms of local heat-transfer coefficient.
Results from the liquid-crystal heat-transfer experiment are illustrated in this figure. The different colors on the liquid-crystal surface correspond to different surface temperatures, which, when combined with the known level of surface heat input, lead to experimentally determined values of surface heat-transfer coefficient. Regions of high and low heat transfer are pointed out in the figure. By photographing the surface and then digitizing the photographic image, quantitative values of surface heat-transfer coefficient are obtained over the entire surface.
NORMAL SHOCK - BOUNDARY-LAYER RESEARCH

Shock - boundary-layer interaction research will be illustrated by describing the normal shock - boundary-layer interaction research program.

INLETS, DUCTS, AND NOZZLES
FLOW PHENOMENA

- HIGHLY 3D FLOW FIELDS
  - TRANSITION DUCTS
  - OFFSET DUCTS

- SHOCK-BOUNDARY-LAYER INTERACTIONS
  - NORMAL SHOCK-BOUNDARY LAYER
  - OBLIQUE SHOCK-BOUNDARY LAYER
  - GLANCING SIDEWALL SHOCK-BOUNDARY LAYER

- SHEAR LAYER CONTROL
  - VORTEX GENERATORS
  - BOUNDARY-LAYER BLEED
  - ENHANCED JET MIXING
This figure illustrates the nature of the normal shock-boundary-layer interaction. The interaction is shown at two Mach numbers, 1.3 and 1.6, and the nature of the interaction is illustrated with a schlieren photograph and a surface oil streak photograph at each Mach number. The photographs were taken in the Lewis 1- by 1-ft supersonic wind tunnel. The schlieren photographs give a view of the interaction integrated across the full width of the test section, while the surface oil streaks illustrate the details of the flow field along the side wall of the test section. At both Mach numbers the schlieren photographs show the shock being bifurcated into a "lambda shock" in the interacting region. The sidewall surface oil streaks indicate a significant difference in the structure of the flow field for the two cases. At Mach 1.3 the flow along the sidewall remains uniform and passes through the shock structure with no major alteration. At Mach 1.6 the adverse pressure gradient across the shock is strong enough to force the boundary layer to separate from the tunnel walls and to form the closed separation bubble shown in the figure. Note that this is a two-dimensional slice through a highly three-dimensional flow field that exists in the corner region.
NORMAL SHOCK - BOUNDARY-LAYER INTERACTION -
LASER ANEMOMETRY RESULTS

Both the Mach 1.3 and 1.6 shock - boundary-layer interaction flow fields were surveyed in detail using nonintrusive laser anemometry. Results are shown for the Mach 1.6 case in two planes within the flow field. The top set of Mach contours illustrates the nature of the flow field in a cross plane downstream of the shock. The lower set of Mach contours shows the development of the flow field within a vertical plane passing through the centerline of the test section. The separated region in the vicinity of the initial interaction causes the actual flow area to contract downstream of the initial shock, leading to a reacceleration of the flow to supersonic Mach numbers. The flow then shocks down again, reaccelerates again due to the thickened boundary layer, and finally shocks down a final time.

RESULTS OF LASER ANEMOMETER MEASUREMENTS
FOR NORMAL SHOCK-Boundary Layer
INTERACTION AT MACH NUMBER 1.6

HALF SECTION SHOWING TYPICAL MEASUREMENT PLANES

MACH NUMBER CONTOURS FROM MEASUREMENTS MADE ON CROSS-PLANE DOWNSTREAM OF SHOCK

MACH NUMBER CONTOURS FROM MEASUREMENTS MADE ON AXIAL MID-PLANE OF TUNNEL
The lower portion of the previous figure is repeated as the lower portion of this figure, that is, Mach contours measured with laser anemometry in a vertical plane passing through the centerline of the test section at Mach 1.6. The upper portion of the figure is a two-dimensional Navier-Stokes computation of the same flow field. Although the computation captures fairly well the initial portions of the flow in the vicinity of the first shock, in the downstream region, none of the flow physics are adequately represented. This result is, of course, expected since the experimental results have shown the strong three-dimensional character of the flow. This comparison points out the need for three-dimensional computational methods for computing such flow fields.
This figure is a frame from a film that will be shown during the conference. The film illustrates the additional insight and understanding that can be obtained from setting in to motion a three-dimensional contour representation of the flow field. The graphics software used to generate the film was actually developed for presenting computational results. The experimental laser anemometry data sets were modified to a format that was acceptable to the graphics software. The researcher can rotate the image, adjust the rate of rotation, and select the axis of rotation while sitting at the display console. One gains a perspective from these rotating images much more quickly than one would by looking at a series of two-dimensional or even three-dimensional plots.
The first aspect of shear-layer control, boundary-layer control, will be illustrated by describing the vortex generator research program.

INLETS, DUCTS, AND NOZZLES
FLOW PHENOMENA

• HIGHLY 3D FLOW FIELDS
  - TRANSITION DUCTS
  - OFFSET DUCTS

• SHOCK-BOUNDARY-LAYER INTERACTIONS
  - NORMAL SHOCK-BOUNDARY LAYER
  - OBLIQUE SHOCK-BOUNDARY LAYER
  - GLANCING SIDEWALL SHOCK-BOUNDARY LAYER

• SHEAR LAYER CONTROL
  - VORTEX GENERATORS
  - BOUNDARY-LAYER BLEED
  - ENHANCED JET MIXING
This figure illustrates a research model and facility that were used in an experiment to assess the performance of vortex generators in a diffusing offset duct. The duct had a length-to-diameter ratio of 5.0, an offset-to-diameter ratio of 1.34, and an exit-to-inlet area ratio of 1.50. Initial experiments with the duct identified the location of a separated flow region as shown by the surface oil streak photograph. Vortex generators were then added to the duct surface just upstream of the separated region to control the separation.
A comparison of total-pressure contours at various locations down the length of the duct with the vortex generators in place, is shown in this figure. The experimental data were obtained by surveying the flow field with a total-pressure probe. The computational results are from the three-dimensional PNS method. The method includes a model for the vortex generators that allows one to position the individual generators anywhere within the duct and permits an adjustment to the strength of the vortex based on an empirical relationship with the vortex generator angle of attack.

In order to illustrate more clearly the influence of the vortex generators, a film will be shown of the total-pressure contours that one would see while moving downstream through the duct. Two segments of film will be shown—the first without the vortex generators and the second with the vortex generators.
Another aspect of the shear layer control, the use of aerodynamic excitation to control the development and evolution of large-scale structures in a shear layer, will be illustrated by describing the enhanced jet mixing research program.

INLETS, DUCTS, AND NOZZLES
FLOW PHENOMENA

• HIGHLY 3D FLOW FIELDS
  - TRANSITION DUCTS
  - OFFSET DUCTS

• SHOCK-Boundary-LAYER INTERACTIONS
  - NORMAL SHOCK-Boundary LAYER
  - OBLIQUE SHOCK-Boundary LAYER
  - GLANCING SIDEWALL SHOCK-Boundary LAYER

• SHEAR LAYER CONTROL
  - VORTEX GENERATORS
  - BOUNDARY-LAYER BLEED
  - ENHANCED JET MIXING
The mixing process between a jet and the surrounding air flow can be enhanced through use of aerodynamic excitation. The naturally occurring flow structure of the jet mixing process is shown in the left-hand Schlieren photograph. By applying an excitation signal at the proper frequency, here by use of acoustic drivers upstream of the jet exit plane, the naturally occurring large-scale structures within the mixing layer are regularized and enhanced and lead to a more rapid mixing process as illustrated in the center photograph.
Results of an axisymmetric Navier-Stokes computation of a jet flow exiting into a quiescent region are shown in this figure. The jet exit Mach number is 0.3, and the results are shown in terms of vorticity contours. Two cases are shown – on the left side an unexcited case and on the right side an excited case. For the excited case the excitation signal is applied at a frequency chosen to maximize the development and growth of the large-scale structures. In both cases results of the computation are shown at three different times. Also for both cases lines are drawn through the centers of the large-scale vortices to illustrate how they propagate downstream. Note that, with excitation applied at the proper frequency, adjacent vortices combine or pair to form a single-larger vortex, which, in turn, has the effect of more rapidly mixing the jet flow with the surrounding quiescent flow.
The effect of excitation on the jet mixing process for a jet having an exit Mach number of 0.3 and an initial turbulence level of 0.15 percent is shown in this figure. The results were obtained in an experiment wherein both the frequency and the level of the excitation signal could be varied. The results show how the ratio of centerline velocity for the excited case to that of the unexcited case varies as the level of the excitation is increased at a fixed frequency. The measurements are made nine jet diameters downstream of the nozzle exit plane. As in the previous figure, the excitation frequency has been selected to provide maximum mixing enhancement. As indicated, the effect of the excitation is quite significant with, in this case, a reduction in centerline velocity of about 16 percent at the maximum available excitation level of 130 dB.
In summary, the internal fluid mechanics research program in inlets, ducts, and nozzles is a balanced effort between the development of computational tools and the conduct of experimental research. The program has been briefly described by highlighting research efforts in highly three-dimensional flow fields, shock-boundary-layer interactions, and shear-layer control. Much of the computational focus until now has been on the development and validation of parabolized Navier-Stokes methods. More recently and in the future, more emphasis will be placed on the development and validation of three-dimensional Navier-Stokes methods. The experimental element of the program will continue to provide a fundamental understanding of the fluid flow physics, to develop new and/or improved flow models, and to provide benchmark data-sets for computational method validation of both the parabolized and full Navier-Stokes methods.

### INLETS, DUCTS, AND NOZZLES
#### SUMMARY

<table>
<thead>
<tr>
<th>COMPUTATIONAL METHODS</th>
<th>EXPERIMENTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>3D PARABOLIZED NAVIER-STOKES (PNS)</td>
<td>(1) TRANSITION DUCTS</td>
</tr>
<tr>
<td></td>
<td>(2) OFFSET DUCTS</td>
</tr>
<tr>
<td></td>
<td>(3) SHOCK-BOUNDARY-LAYER INTERACTION</td>
</tr>
<tr>
<td></td>
<td>(4) VORTEX GENERATORS</td>
</tr>
<tr>
<td></td>
<td>(5) BOUNDARY-LAYER BLEED</td>
</tr>
<tr>
<td>3D NAVIER-STOKES (NS)</td>
<td>(6) EXPERIMENTS (1) TO (5)</td>
</tr>
<tr>
<td></td>
<td>(7) SEPARATION FLOW PHYSICS</td>
</tr>
<tr>
<td></td>
<td>(8) NONORTHOGONAL SURFACES</td>
</tr>
<tr>
<td></td>
<td>(9) JET MIXING</td>
</tr>
</tbody>
</table>

CD-87-29873

3-28
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.

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.
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.
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.
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.
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.
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.

**AXIAL MOMENTUM EQUATION:**

\[
\frac{\partial}{\partial t} \left( \lambda_1 \dot{r} V + \frac{\partial}{\partial \varphi} \lambda_1 \dot{r} \dot{V} V + \frac{\partial}{\partial \varphi} \lambda_1 \dot{r} \dot{V} V + \frac{\partial}{\partial \varphi} \lambda_1 \dot{r} \dot{V} V + \frac{\partial}{\partial \varphi} \right) \left( \dot{r} \dot{V} V + \ddot{V} \right)
\]

Convective Terms

\[
= \frac{\partial}{\partial \rho} \lambda_1 \left( \dot{r} \dot{V} V \right) + \frac{\partial}{\partial \rho} \lambda_1 \left( \dot{r} \dot{V} V \right) + \frac{\partial}{\partial \rho} \lambda_1 \left( \dot{r} \dot{V} V \right) + \frac{\partial}{\partial \rho} \lambda_1 \left( \dot{r} \dot{V} V \right) + \frac{\partial}{\partial \rho} \lambda_1 \left( \dot{r} \dot{V} V \right)
\]

Diffusive Terms

\[
+ \frac{\partial}{\partial \rho} \lambda_1 \left( \dot{r} \dot{V} V \right) + \frac{\partial}{\partial \rho} \lambda_1 \left( \dot{r} \dot{V} V \right) + \frac{\partial}{\partial \rho} \lambda_1 \left( \dot{r} \dot{V} V \right) + \frac{\partial}{\partial \rho} \lambda_1 \left( \dot{r} \dot{V} V \right)
\]

Body Force Terms

\[
+ \frac{\partial}{\partial \rho} \lambda_1 \left( \dot{r} \dot{V} V \right) + \frac{\partial}{\partial \rho} \lambda_1 \left( \dot{r} \dot{V} V \right) + \frac{\partial}{\partial \rho} \lambda_1 \left( \dot{r} \dot{V} V \right) + \frac{\partial}{\partial \rho} \lambda_1 \left( \dot{r} \dot{V} V \right)
\]

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 unknowns which have to be determined by conducting physical and/or numerical experiments.
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.
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.
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.
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.
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.
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.
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

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

CD-87-24905

ORIGNAL PAGE IS
OF POOR QUALITY

3-46
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.
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.
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 assessment 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.
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.
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.
CHEMICAL REACTING FLOWS
Edward J. Mularz
and
Peter M. Sockol

ABSTRACT
Future aerospace propulsion concepts involve the combustion of liquid or gaseous fuels in a highly turbulent internal air stream. Accurate predictive computer codes which can simulate the fluid mechanics, chemistry, and turbulence-combustion interaction of these chemical reacting flows will be a new tool that is needed in the design of these future propulsion concepts. Experimental and code development research is being performed at Lewis to better understand chemical reacting flows with the long term goal of establishing these reliable computer codes.

Our approach to understanding chemical reacting flows is to look at separate more simple parts of this complex phenomena as well as to study the full turbulent reacting flow process. As a result we are engaged in research on the fluid mechanics associated with chemical reacting flows. We are also studying the chemistry of fuel-air combustion. Finally, we are investigating the phenomena of turbulence-combustion interaction. This presentation will highlight research, both experimental and analytical, in each of these three major areas.
CHEMICAL REACTING FLOWS

Chemical reacting flows of aerospace propulsion systems have features similar to other internal flows described previously. The flows are typically highly turbulent, with large secondary flows and three-dimensional flow characteristics. Flow oscillations and unsteadiness is usually noticed in these flows. There are additional features which are unique to flows with combustion which adds a great deal of complexity to the process. This includes a substantial increase in temperature as the flow moves downstream, and a significant change in fluid properties due to fluid species changes. In addition, the time scale for combustion is often orders of magnitude different than the fluid flow time, and there is often a strong interaction between the turbulent flow and the combustion process. These complex features not only make experimental studies difficult but also significantly affect the methods for computing these flows with computer codes.

CHEMICAL REACTING FLOWS

HAS FEATURES OF INTERNAL FLOWS:

- TURBULENT
- HIGHLY THREE-DIMENSIONAL
- FLOW UNSTEADINESS

HAS ADDITIONAL COMPLEXITY:

- SUBSTANTIAL HEAT RELEASE
- MANY CHANGE OF SPECIES
- COMBUSTION TIME << FLOW TIME SCALE
- FLOW—COMBUSTION INTERACTION
The research activities in chemical reacting flows are divided into three areas as follows: (1) Fluid Mechanics, which looks at the fluid flow phenomena associated with combustion without the added complexity of including heat release. This includes the multiphase processes of fuel sprays, and the highly three-dimensional and time varying flows that typically exist in real propulsion systems. (2) Combustion Chemistry, which concentrates on the combustion of fuel and oxidizer without including the fluid mechanics. Research is being done to understand the ignition process of fuel and oxidizer and to probe the detailed chemistry to obtain an accurate combustion model for further fluid codes. Catalytic combustion is also being studied as a fuel processor for high speed propulsion. (3) Turbulence Combustion Interaction, which looks at both the fluid mechanics and the chemistry of combustion and their effects on each other. Work is being done to understand the key features of turbulent reacting flow and to construct accurate computer codes to simulate this flow. As a useful "numerical experiment," the technique of direct numerical simulation is also being used to better understand chemical reacting flows.

These three major areas of activities are all being performed to achieve the long term goal to obtain an accurate predictive code with coupled fluid mechanics and chemistry which will be helpful in the design of future aerospace propulsion systems.

Let us now look at an example of the research in the area of Fluid Mechanics research on multiphase flow of liquid fuel sprays in air.
MULTIPHASE FLOWS

The fuel-spray process is extremely important in terms of engine efficiency, durability, and operability. The ultimate objective of the research is to develop a computer code that can accurately model the fuel and air mixing with subsequent combustion process. Since this is a very complicated process, it is being approached in a series of steps of increasing complexity. Particle-laden jets were initially studied in order to assess the capability of current two-phase flow models. Evaporating liquid sprays and then combusting sprays will also be studied.

MULTIPHASE FLOWS

OBJECTIVE:
DEVELOP COMPUTER CODE WHICH ACCURATELY MODELS FUEL SPRAY/AIR MIXING AND COMBUSTION PROCESSES

APPROACH:

(1) PARTICLE-LADEN JET EXPERIMENT
(2) TWO-PHASE FLOW COMPUTER MODELS

CD-87-28756
This photograph illustrates the experimental arrangement of the particle-laden jet. An air jet containing solid glass beads (39 micron, Sauter Mean Diameter) discharged downward into a still environment. Particle-laden jets with three swirl numbers were studied. Nonintrusive measurements of velocity were obtained with a two-channel laser velocimeter. Particle size and velocity were measured with a phase/doppler particle anemometer. The gas phase was seeded with nominal 1-micron diameter aluminum oxide powder to measure gas phase velocities.
MULTIPHASE FLOW VELOCITY COMPARISON

This figure presents typical results from the particle-laden jet study. A contour plot of experimentally measured axial velocity of the gas phase (left side) and particle phase (right side) is illustrated. It is evident that initially, the gas phase has a higher velocity than the particle phase. The particles are initially accelerated by the gas phase and then their velocity begins to decay. Because of their inertia, the rate of decay of axial velocity is slower for the particles than the gas. Also shown in the figure are predictions from the SSF model at 10 diameters downstream of the tube exit. This model tracks particle trajectories in the computed gas phase flowfield and allows momentum exchange between phases. The model also considers effects of gas-phase turbulence on particle trajectories. Predictions from the model show reasonable agreement with the data.

MULTIPHASE FLOW VELOCITY COMPARISON

- PARTICLE DIAMETER, 39 MICRON (SMD)

DATA MODEL

PARTICLES

VELOCITY, U (M/S)

GAS

EXPERIMENTAL DATA

CD-87-28758

ORIGINAL PAGE IS OF POOR QUALITY
Future directions for multiphase flow research include evaporating liquid sprays and combusting liquid sprays. Evaporating sprays are currently being studied under contract at the University of California, Irvine, and Allison Gas Turbine as part of the HOST Program. The test cell where the particle-laden jets were studied is currently being modified to study liquid sprays.

FUTURE DIRECTIONS:

(1) EVAPORATING LIQUID SPRAY
(2) LIQUID FUEL SPRAY COMBUSTION
Let us now turn our attention to the area of Combustion Chemistry. An example of the research in this area is the study of the chemical kinetics of hydrogen-air combustion.
In a supersonic ramjet propulsion system, the time required between the injection of fuel into the airstream and its combustion point is very important with regards to the length of the engine and its weight. These high speed combustion concepts will be tested in wind tunnels where the air has been heated to simulate the aerodynamic heating of the vehicle in the atmosphere. Research is underway to determine the combustion delay time of hydrogen fuel and air and to determine the effects of air contaminants in the wind tunnel on this combustion delay time.

Stoichiometric H₂-O₂ ignition delay times were measured behind reflected shock waves at 1.1 atm pressure over the temperature range 1300 to 950 K by using a chemical shock tube. The proposed chemical kinetic model predicted ignition delay times in excellent agreement with the experimental data.

**CHEMICAL KINETICS**

**OBJECTIVE:**
Determine combustion delay time of hydrogen fuel—air chemistry

**APPROACH:**

(1) Chemical shock tube

(2) Reaction rates computer model

CD-87-28760
The information which we are seeking is to determine how far from the fuel injection point will a stable flame exist in a supersonic airstream of various Mach numbers.

Also, since heated wind tunnels have small amounts of contaminants or additives in the air, we are studying the effects of these contaminants on this combustion delay time.

**SCRAMJET ENGINE SCHEMATIC:**

- HOW FAR WILL STABLE FLAME BE FROM FUEL INJECTION POINT?
- WHAT IS EFFECT OF AIR CONTAMINANTS OR ADDITIVES ON THIS COMBUSTION DELAY?
Shown here are the levels of four major air contaminants which exist in the two U.S. hypersonic wind tunnels when they are simulating a Mach 7 flight speed. Carbon dioxide and water vapor are known to lengthen the combustion delay time. Nitric oxide and nitrous oxide although orders of magnitude smaller in concentrations, would tend to shorten the combustion delay time.

HEATED WIND TUNNEL AIR CONTAMINANTS

<table>
<thead>
<tr>
<th>Contaminant</th>
<th>Concentration</th>
</tr>
</thead>
<tbody>
<tr>
<td>Carbon Dioxide</td>
<td>6 to 10 percent</td>
</tr>
<tr>
<td>Water Vapor</td>
<td>13 to 17 percent</td>
</tr>
<tr>
<td>Nitric Oxide</td>
<td>0.9 percent</td>
</tr>
<tr>
<td>Nitrous Oxide</td>
<td>0.025 percent</td>
</tr>
</tbody>
</table>

CD-87-287762
CHEMICAL KINETICS

Applying the chemical kinetics computer model to the prediction of combustion delay time for a scramjet flying at a Mach 7 flight condition predicted a 70 cm distance between the fuel injection point and the flame front. When the wind tunnel contaminants were taken into account, this distance was significantly shorter: only 13 cm between the fuel injection point and the flame front. Thus, the small concentration of contaminants (nitric oxide and nitrous oxide) resulted in over a factor of five reduction in combustion length. Not only is this effect important in evaluating wind tunnel experiments of combustion concepts, but it also indicates that trace additives into the flow could significantly shorten the required engine length and thereby considerably reduce weight.

This work is continuing to further explore the effects of these trace contaminants or additives and to better quantify their potential benefit to future scramjet designs.

CHEMICAL KINETICS

CALCULATED COMBUSTION DELAY FOR MACH 7 SCRAMJET FLIGHT CONDITION:

(A) BASED ON PURE DRY AIR.

(B) BASED ON PURE AIR WITH CONTAMINANTS OR ADDITIVES.
While it is important and quite useful to look at the fluid mechanics and the combustion chemistry aspects of chemical reacting flows independently, to get a full understanding of the dominant phenomena of these flows, we must examine the interaction of turbulent flow with combustion. The activities in this area include both numerical code development work and experimental research. We will now highlight the research in turbulent reacting flow.

### CHEMICAL REACTING FLOWS

<table>
<thead>
<tr>
<th>FLUID MECHANICS</th>
</tr>
</thead>
<tbody>
<tr>
<td>• MULTIPHASE FLOW</td>
</tr>
<tr>
<td>• COHERENT STRUCTURES</td>
</tr>
<tr>
<td>• HIGHLY 3-D FLOWS</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>COMBUSTION CHEMISTRY</th>
</tr>
</thead>
<tbody>
<tr>
<td>• CHEMICAL KINETICS</td>
</tr>
<tr>
<td>• DIFFUSION FLAMES, PREMIXED FLAMES</td>
</tr>
<tr>
<td>• CATALYTIC COMBUSTION</td>
</tr>
</tbody>
</table>

**TURBULENCE—COMBUSTION INTERACTION**

- TURBULENT REACTING FLOWS
- DIRECT NUMERICAL SIMULATIONS

**LONG TERM GOAL:** ACCURATE PREDICTIVE CODE WITH COUPLED FLUID MECHANICS AND CHEMISTRY FOR FUTURE AEROSPACE PROPULSION

CD-87-28755
The objective of this work is to understand the coupling between fluid dynamics and combustion and to establish an accurate computer code which simulates the dominant features of turbulent reacting flow. An experiment is being constructed to focus on many of these features, both steady state and unsteady. This experiment is to examine a plane reacting free shear layer. Turbulent reacting flow computer codes, both steady state and time accurate, are also being developed concurrently, and the database from the experiment will serve as a means to validate these new computer codes.
The objectives of the Planar Reacting Shear Layer Experiment, shown as a schematic here, is to (1) understand the coupling between fluid dynamics and combustion and (2) to establish a data set to validate computer codes which simulate the physics and chemistry of high speed chemical reacting flow. Two gas streams, one of hydrogen and nitrogen, the other of air, will mix downstream of a plane splitter plate. Combustion will occur where the fuel and air has properly mixed. Pressure oscillations in this closed duct will exist due to the dynamic features of the flow and the interactions between these pressure oscillations and the combusting shear layer will be examined. The unique features of this experiment are (1) a continuous flow capability, (2) flow velocities of both the air and the hydrogen-nitrogen mixture which are in the high subsonic range, (3) the air will be heated ahead of the mixing shear layer without any contamination effects, and (4) the heat release in this experiment will be quite high, typical of propulsion systems. The experiment is in fabrication and nonintrusive instrumentation is being purchased. Experiments are expected to begin in spring 1988.
PROMETHEUS II is a time-accurate version of a two-dimensional, finite volume TEACH code. Second-order accurate QUICK differencing is used for the convective terms and block-correction combined with Stone's strongly implicit algorithm is used to solve the pressure correction equation. This results in a highly efficient computational tool for performing "numerical experiments."

TURBULENT REACTING FLOW

TIME-ACCURATE, 2-D SHEAR LAYER CODE

- TWO-DIMENSIONAL NAVIER-STOKES EQUATIONS SOLVED WITH OR WITHOUT A TWO-EQUATION TURBULENCE MODEL
- SECOND-ORDER ACCURACY IN BOTH TIME AND SPACE
- FULLY IMPLICIT NUMERICAL SCHEME DERIVED FROM "SIMPLE" (SEMI-IMPLICIT PRESSURE LINKED EQUATIONS) ALGORITHM
Shown are vorticity contours for a two-dimensional, numerical calculation of a forced, shear layer at a Reynolds number of about 100,000. The positive and negative vorticity contours originate at the boundary layers, specified at the inlet of the computational domain. Forcing is applied at a long wavelength and smaller scale vorticities spontaneously develop as a result of the natural instability of the layer. These small scale vorticities cluster on the scale of the longer, forced wavelength. Small pockets of positive vorticity persist as remnants of the low speed boundary layer. The collective interaction of these small scale vortices, merging into larger scale structures, largely controls the dynamics of the shear layer. These calculations were performed on the NAS Cray 2 computer.
The RPLUS2D code is being developed for calculation of mixing and chemical reactions in the flow fields of ramjets and scramjets. The code is written in generalized curvilinear coordinates and therefore can handle any arbitrary two-dimensional geometry. The implicit LU (lower-upper) scheme used in the RPLUS2D code requires only scalar diagonal inversions while most other implicit schemes require block matrix inversions. The use of scalar diagonal inversions offers order-of-magnitude efficiency improvement over conventional implicit schemes when large systems of partial differential equations must be solved, such as flows in ramjet and scramjet engines. A three-dimensional version of this code will be developed subsequent to the completion of this two-dimensional code.

**TURBULENT REACTING FLOW**

**STEADY STATE FLUID MECHANICS COMPUTER CODE:**

**IMPLICIT L U SYMMETRIC GAUSS-SEIDEL ALGORITHM WITH FINITE VOLUME DISCRETIZATION**

- MAJOR ADVANTAGE OF ELIMINATING BLOCK INVERSIONS WHERE THE NUMBER OF BLOCKS COULD BE QUITE LARGE FOR REACTING FLOWS
- REAL GAS PROPERTIES
- KINETIC REACTION RATE SOURCE TERMS TREATED IMPLICITLY
The $\text{H}_2$ jet injected at right angles to the main airflow acts essentially as an effective body which blocks the flow resulting in a bow shock and strong pressure gradients on the front side of the jet. The jet shock around the injector can also be seen in this figure. The two bow shocks resulting from the two $\text{H}_2$ jets at top and bottom walls (symmetric about the centerline shown) intersect at the center plane and yield a large pressure rise.
In conclusion, research in the area of Chemical Reacting Flow will lead to an understanding of this complex flow and accurate predictive computer codes. Activity in this topic is focused on three areas: Fluid Mechanics, Combustion Chemistry, and Turbulence-Combustion Interaction. Results from this research will provide important technical "tools we need" to develop new aerospace propulsion systems for the year 2000 and beyond.
16. **Abstract**

Internal fluid mechanics research at Lewis is directed toward an improved understanding of the important flow physics affecting aerospace propulsion systems, and applying this improved understanding to formulate accurate predictive codes. To this end, research is conducted involving detailed experimentation and analysis. The presentations in this session summarize ongoing work and indicate future emphasis in three major research thrusts; namely, inlets, ducts and nozzles, turbomachinery, and chemical reacting flows.

17. **Key Words (Suggested by Author(s))**

Aeronautical propulsion research; Materials; Structures; Internal fluid mechanics; Instrumentation; Controls; Subsonic propulsion; Supersonic propulsion; Hypersonic propulsion