55 5144

NASA LANGLEY RESEARCH CENTER
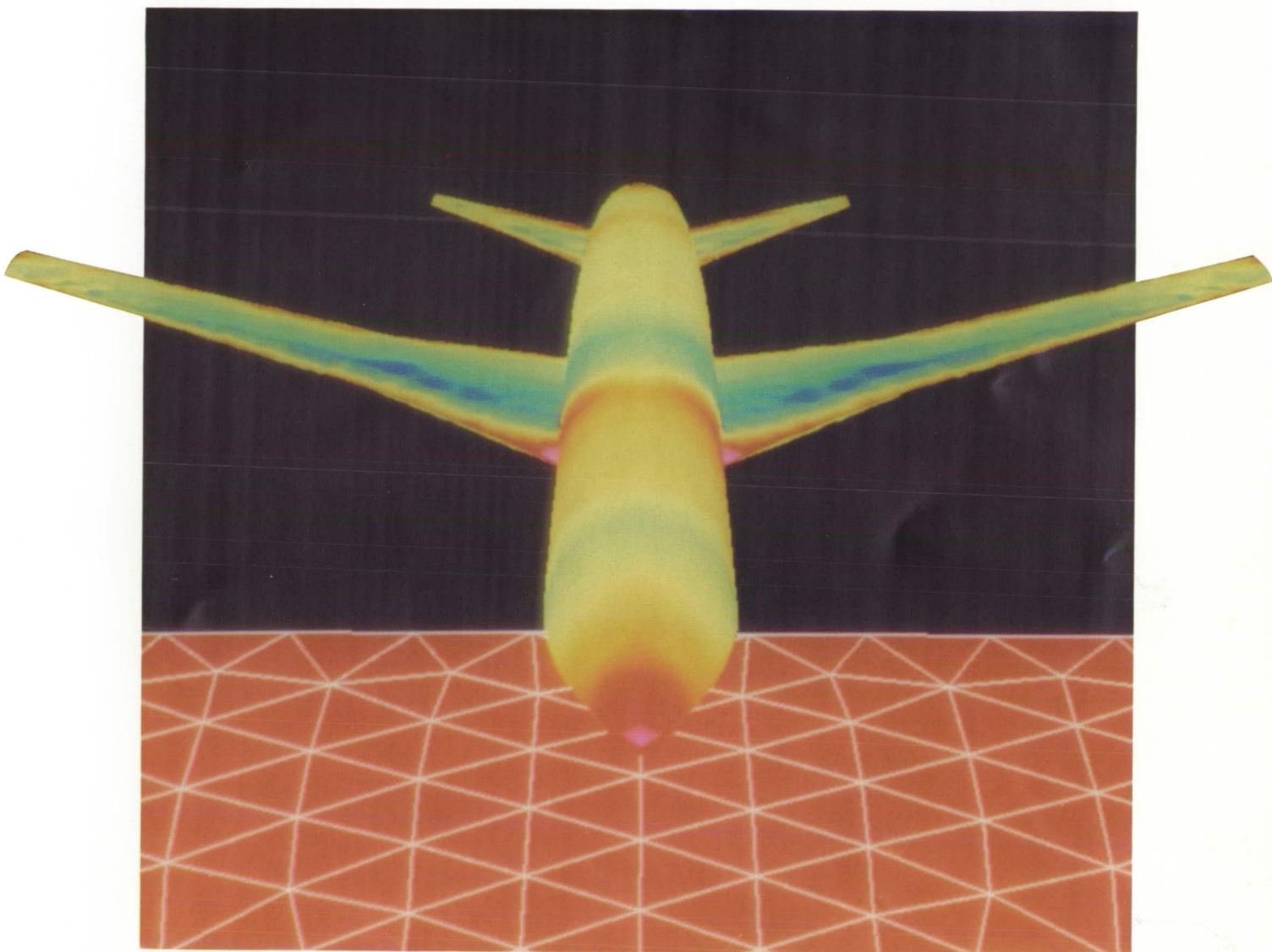
COMPUTATIONAL
FLUID DYNAMICS

The role of Langley Research Center is to perform the basic and applied research necessary for the advancement of aeronautics and space flight, to generate new and advanced concepts for the accomplishment of related national goals, and to provide research advice, technological support, and assistance to other NASA installations, other government agencies, and industry. One of the tools used more and more heavily at Langley, as well as other NASA Centers such as Ames and Lewis, is computational fluid dynamics (CFD). Applying the laws of fluid physics to mathematical equations and solving them with high-speed supercomputers has resulted in an increasingly sophisticated simulation of the forces and moments acting on aircraft and spacecraft in flight. CFD has also gained widespread acceptance as a design tool among aircraft manufacturers and provides important information that often complements and sometimes goes beyond what can be learned from wind tunnel tests alone.

At Langley, the CFD research effort is not concentrated in a single division or other organizational entity, but is distributed throughout the Center. There are some 15 branches in four research directorates which incorporate a significant component of CFD research and advanced applications. This management philosophy at Langley recognizes the crucial importance of CFD to the many aeronautics and space research disciplines and sub-disciplines, as well as the need to keep the CFD research focused on the most urgent problems.

The diversification of Langley's CFD research effort prompts the need for a periodic collection and overview of recent activities. The last such compilation was given in *A Compendium of Computational Fluid Dynamics at the Langley Research Center,* August, 1980. Significant advances in both available computer power and CFD algorithms have occurred over the last decade, so an update is warranted. The present document attempts to provide such an overview. Langley organizations that contributed to this report include the Analytical Methods Branch, Theoretical Aerodynamics Branch, Computational Methods Branch, Unsteady Aerodynamics Branch, Computer Applications Branch, Aerothermodynamics Branch, the Aerothermal Loads Branch, and the Institute for Computer Applications in Science and Engineering (ICASE), an independent institution which is located at Langley. Questions about this report may be directed to the Office of the Chief Scientist at Langley (804) 865-3316.
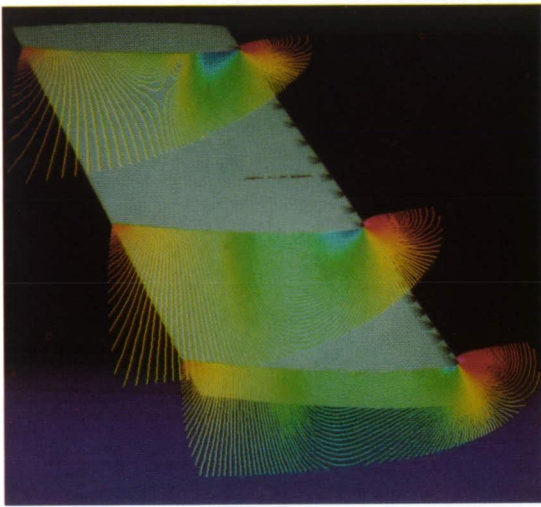
COMPUTATIONAL FLUID DYNAMICS

NASA LANGLEY RESEARCH CENTER

COMPUTATIONAL FLUID DYNAMICS

Over the last six years, NASA Langley has been developing TAWFIVE (Transonic Analysis of a Wing and Fuselage with Interacted Viscous Effects). The program uses two different solutions: one to predict the outer inviscid portion of the flow, the other to compute the inner viscous region. By incorporating multigrid convergence methods, Langley scientists have reduced the required computer time by more than half.

The first figure (right) shows air pressure distributions on the top surface of a wing. The effect of the aircraft's fuselage (not shown) on the wing can be seen in the lower right portion of the wing.

In the second figure, (above), pressure coefficients are plotted along vertical planes that intersect the wing.

The economic and military importance of aerospace research to this country cannot be overestimated. The aerospace industry involves over 15,000 companies, with over a million jobs in all 50 states. Approximately one-third of the $280 billion defense budget goes for aeronautical products and their support. With American aerospace exports in the tens of billions, this industry is the major positive contributor to the trade balance. Because the Japanese and Europeans have made major strides, America's future preeminence in the aerospace industries cannot be assumed. A vigorous, well funded domestic research program must continue.

A crucial factor in the continuation of our world preeminence both economically and militarily is the design of energy efficient and/or high performance aircraft and spacecraft. Langley Research Center is continuing its research programs to provide advanced technology for all vehicle classes, from subsonic to hypersonic.

Langley's CFD program is rapidly becoming an essential element of its research programs in aerodynamics, aerothermodynamics, aeroelasticity, and atmospheric science. The ultimate goal of computational fluid dynamics at Langley is to create a comprehensive mathematical model of airflow around bodies for all speed regimes. What factors in aircraft design can increase the lift and decrease the drag and frictional forces? Why does a smoothly flowing stream of air (called "laminar flow") over an aircraft surface often become completely disordered and chaotic (or "turbulent"), drastically increasing the drag? The onset of turbulence is only crudely predictable today, and even then only in particular situations. The elusive solutions to the basic equations of fluid flow, particularly the "Navier-Stokes equations", are enormously complicated, since they must resolve the ever-changing flows within a turbulent fluid.

Almost from the beginning of the aviation era, wind tunnels with their fan-driven airflows have been employed to obtain measurements of the forces and moments on scaled aerospace models. These measurements are used to gain an understanding of how full-scale aircraft and spacecraft will behave in flight. The first national wind tunnels used for aircraft studies were built at Langley, the original NACA Center (National Advisory Committee
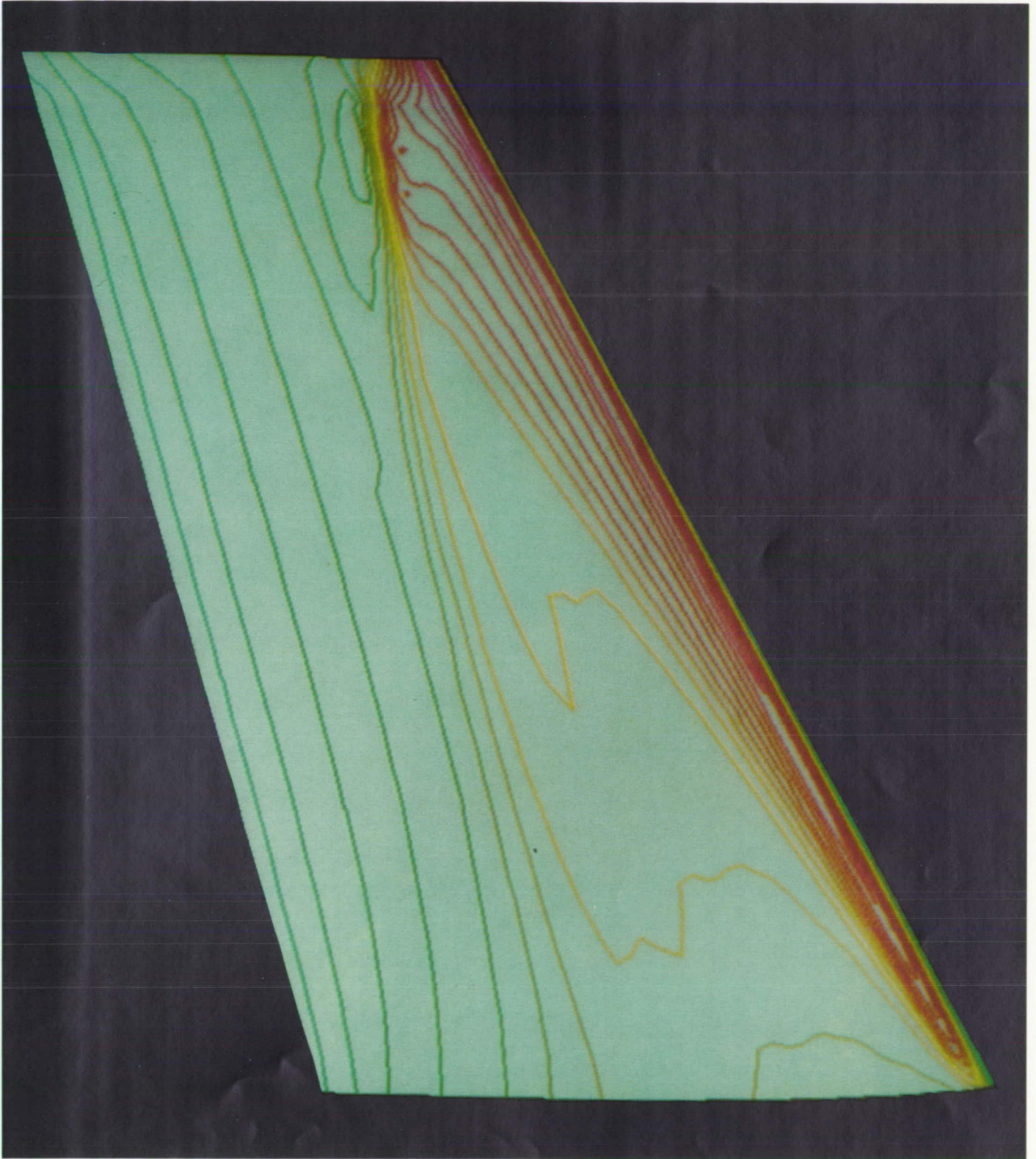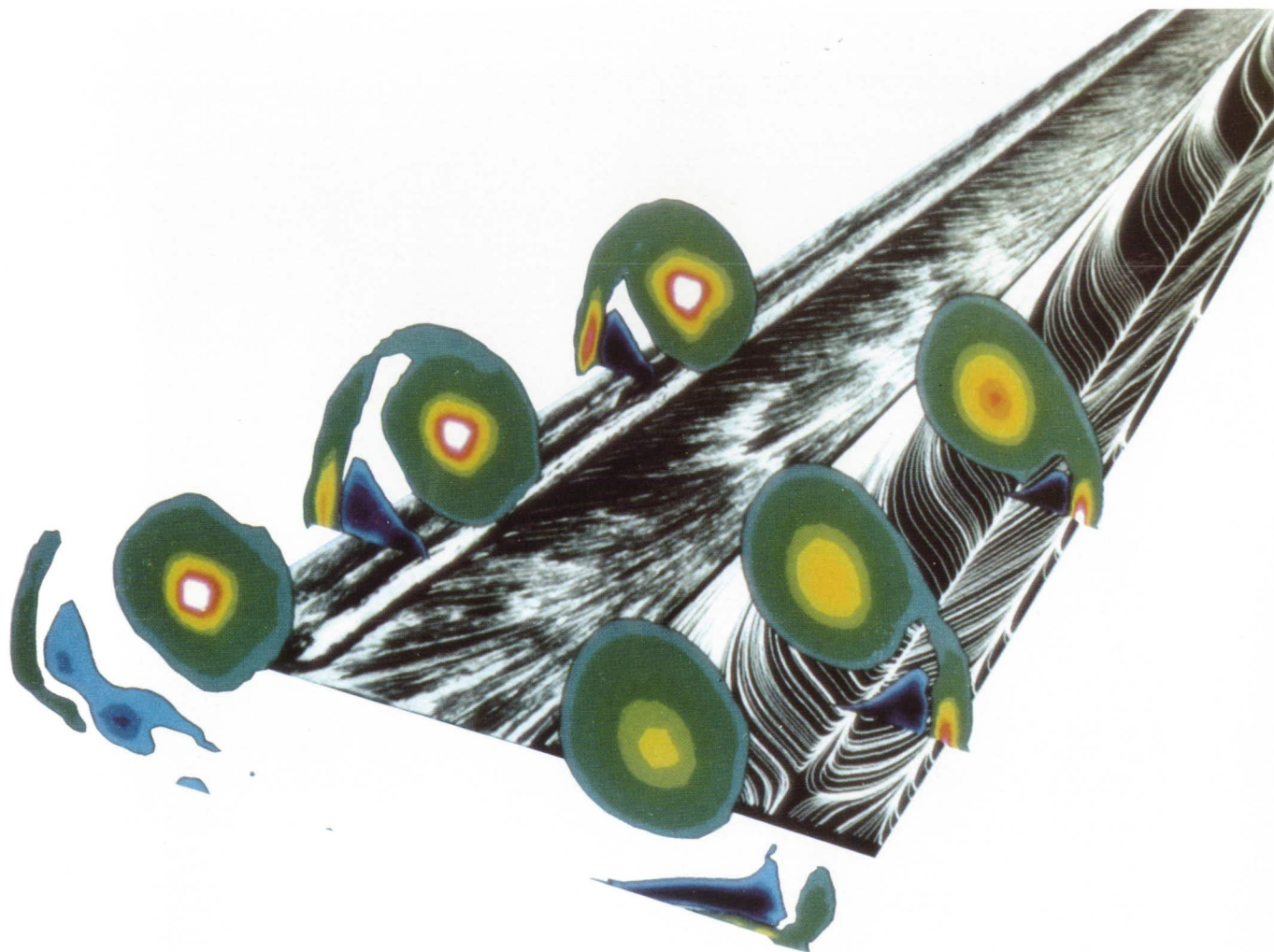
Vorticity contours at three axial locations and surface streamline patterns are compared between experiment (left) and Navier-Stokes computations (right) for the flow over a delta wing at a relatively high angle of attack. This critical information enables Langley scientists to assess future high-performance military aircraft that must be designed to fly and maneuver under the most extreme flight regimes.

for Aeronautics), in 1917. Today, Langley continues that tradition with the largest assortment of wind tunnels in the Free World. Other high quality wind tunnels (at NASA Ames and Lewis Research Centers) also play a major role with Langley in the design and development of modern air and space craft.

CFD has the potential to lower wind tunnel time and costs for design verification, which currently run from $15 to $30 million for a typical major aircraft program. In addition, computationally-designed aerospace components can be design-optimized with the use of an optimization code that performs a computerized "sifting" through a large variety of configurations. The resulting "best" design will then be safer, more energy efficient and/or better performing than one developed without the use of these methods. Once aerospace designers and industry and government managers are confident with CFD, they will find it to be a fast, reliable and economical design tool.
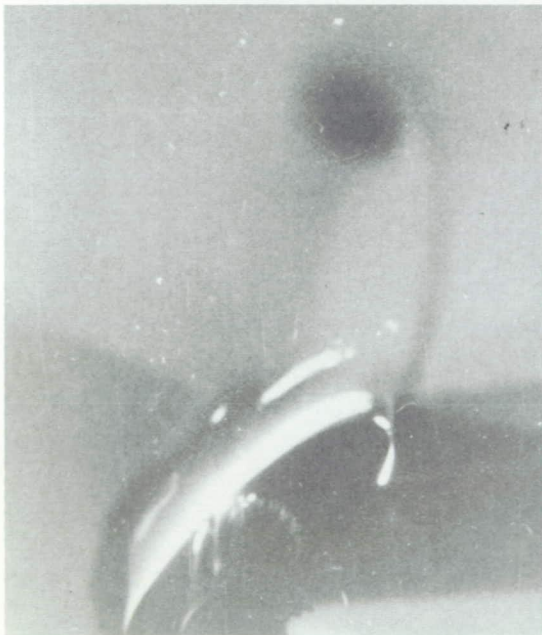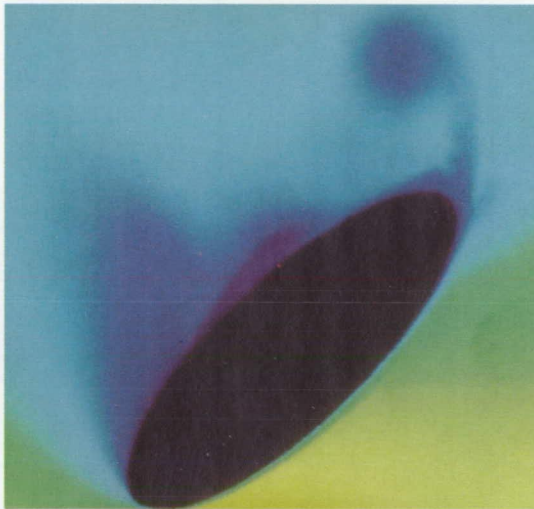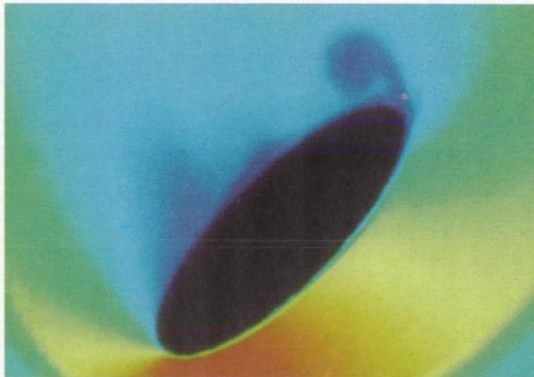
By using an algebraic turbulence model with the Navier-Stokes equations on the Center's VPS-32 supercomputer, NASA Langley scientists can develop mathematical models which approximate the flow physics corresponding to experimental conditions. The vortical flow generated by an elliptical body missile can be seen in this vapor screen photograph (left) taken at M = 2.5 in Langley's Unitary Plan wind tunnel. The second image (right) was generated from a code on the VPS-32 supercomputer; it required only two hours of central processing time.
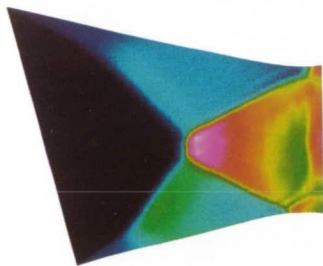
The past 40 years have seen tremendous progress in developing mathematical models of fluid flows, even leading to some controversial suggestions that CFD and wind tunnels are competitive rather than complementary. Langley's approach has always been to apply whatever experimental or computational tool or combination of tools is best suited to understand each individual aerodynamic problem and regime. For example, Langley's research in the "hypersonic" (very high speed) regime uses CFD methods heavily, partly because of the worldwide rarity of experimental capabilities at the highest speeds contemplated for the proposed Aero-Space Plane. On the other hand, Langley employs wind tunnel measurements heavily in the "transonic" (near the speed of sound) regime, partly because of the uncertainties in CFD calcula-
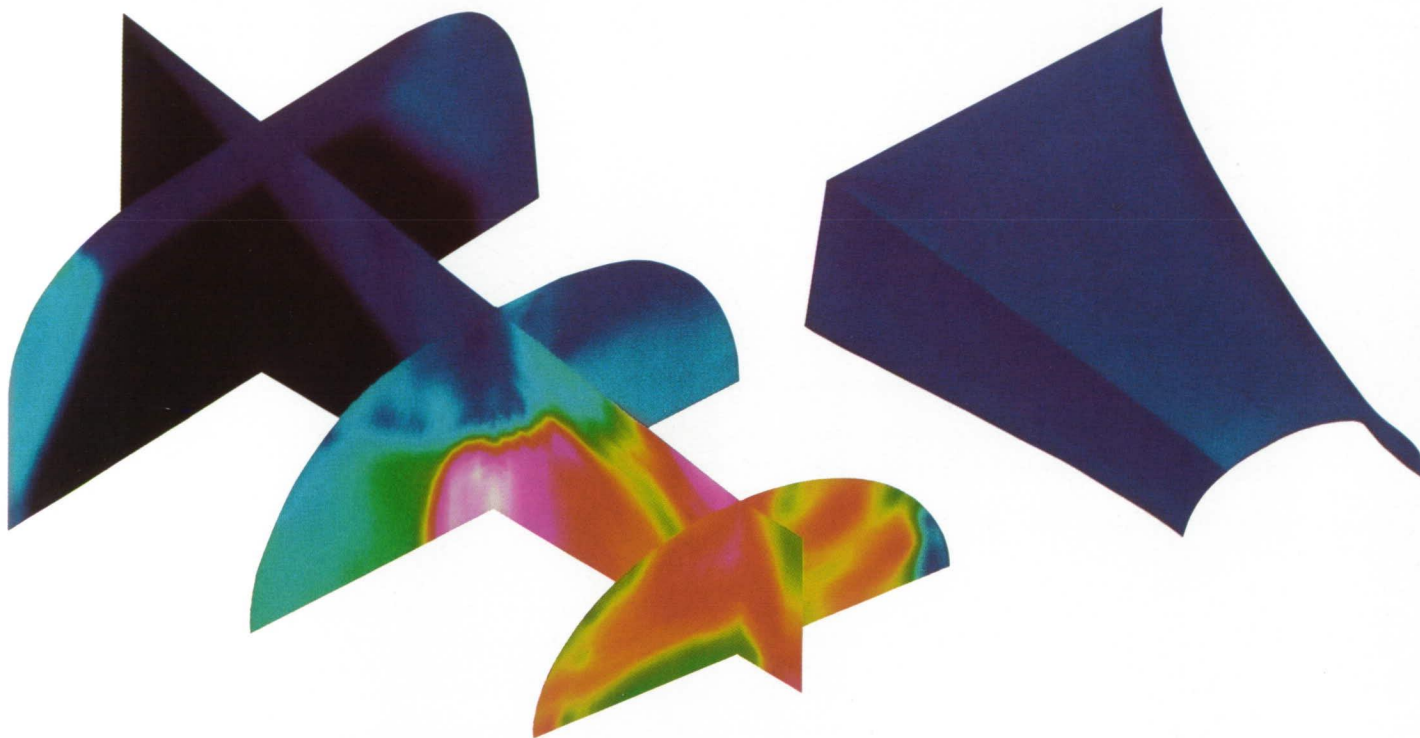
The shape of the scramjet's engine can affect its performance. The intake should be rectangular for integration with the air frame; however, its combustor should be circular for maximum strength and lightness. This image (above) shows the internal pressures (blue low, yellow medium, white high) generated at M = 3.5 when air moves through a proposed inlet design that has a rectangular inlet and circular combustor. These studies greatly reduce the need for building different test configurations.

tions of airflow in the vicinity of shocks which occur at such speeds. Langley's newest wind tunnel, the National Transonic Facility, will help researchers develop more accurate CFD codes for this speed regime.

A major focus of Langley's CFD program has always been the engineering application of computational methods to national aerospace programs. The history of Langley's major CFD applications dates back to the 1960s, including major contributions to the design of the Space Shuttle, the Galileo Probe, certification of the Space Shuttle Thermal Protection System and many others. A striking feature of Langley's current work lies in its emphasis on three-dimensional (3-D) problems. Many laboratories have per-
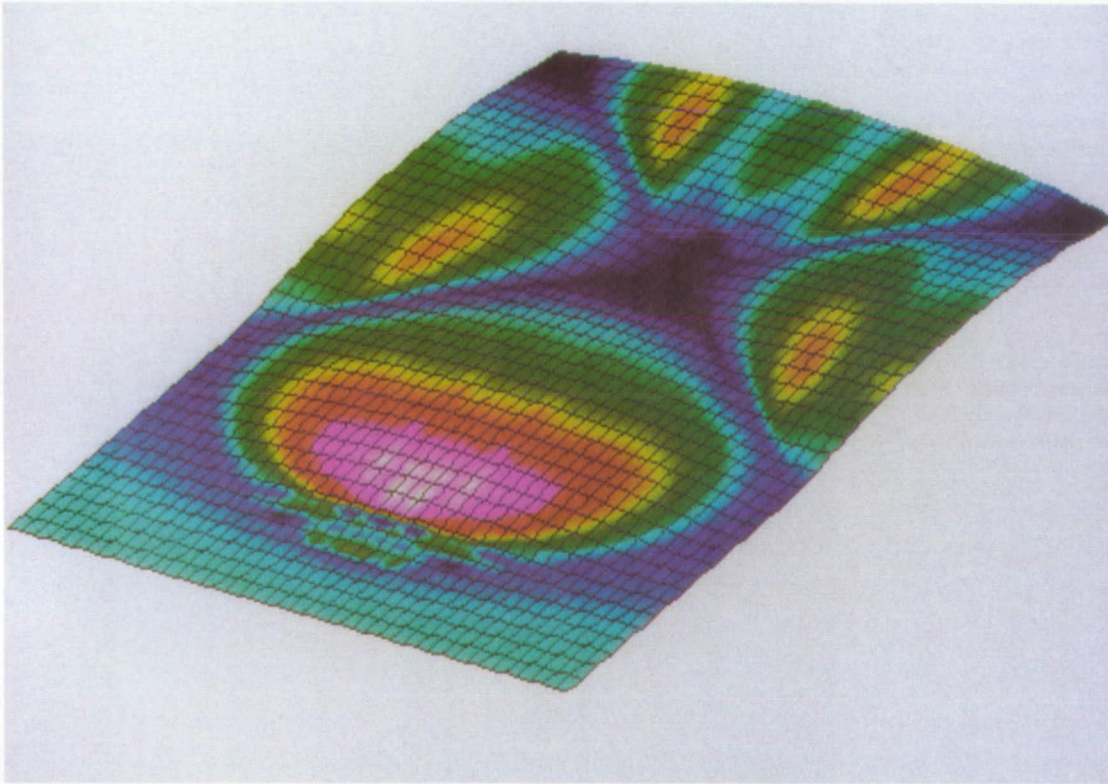
CFD analysis is also being applied in the Space Shuttle program. Researchers have investigated the probable effects of high speed airflow over a proposed metallic thermal tile. A metallic tile will expand and may bow out into the airstream, generating further heat from frictional forces and affecting the air flow. The results, which involve both airflow and thermal loading, show that the proposed tile could be more effective than the ceramic tile currently used.


7
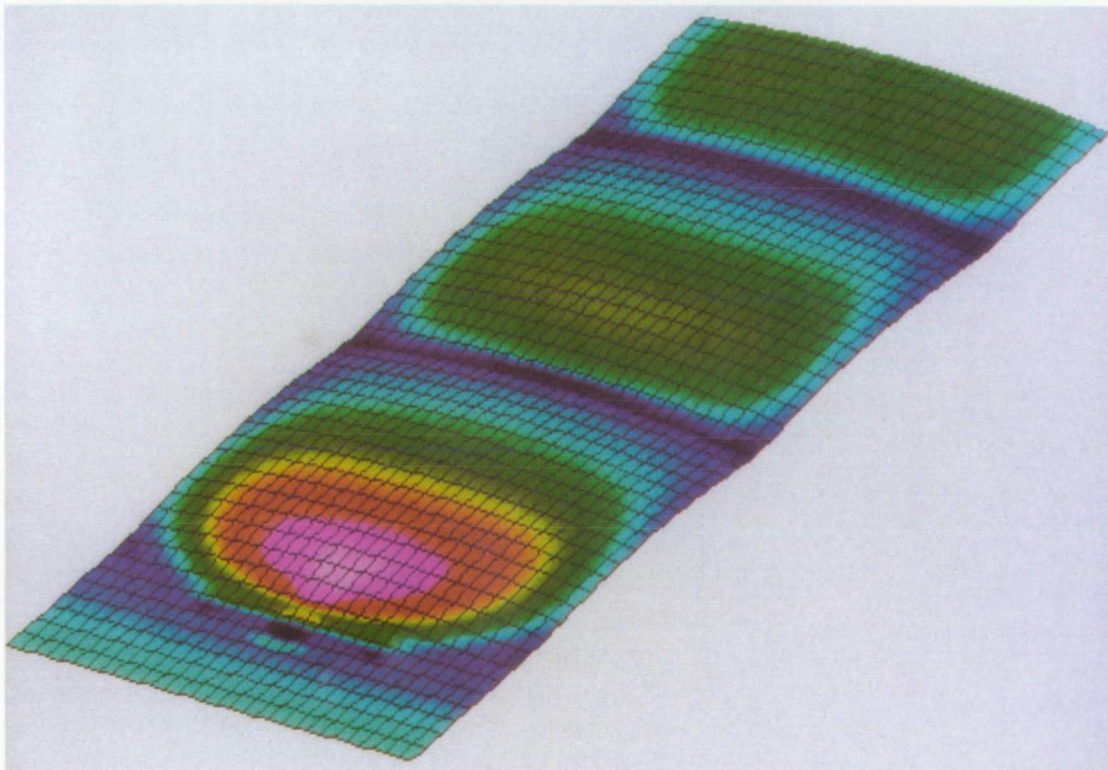
Because no ground-based experimental facilities can fully duplicate conditions encountered in the upper atmosphere, CFD methods become especially important in the study of hypersonic, high altitude flows. These computer-generated images show air pressure (near) and temperature distribution (far) on the Space Shuttle Orbiter at M = 8. High pressures/temperatures are white through pink, and low pressures/ temperatures are black through blue.
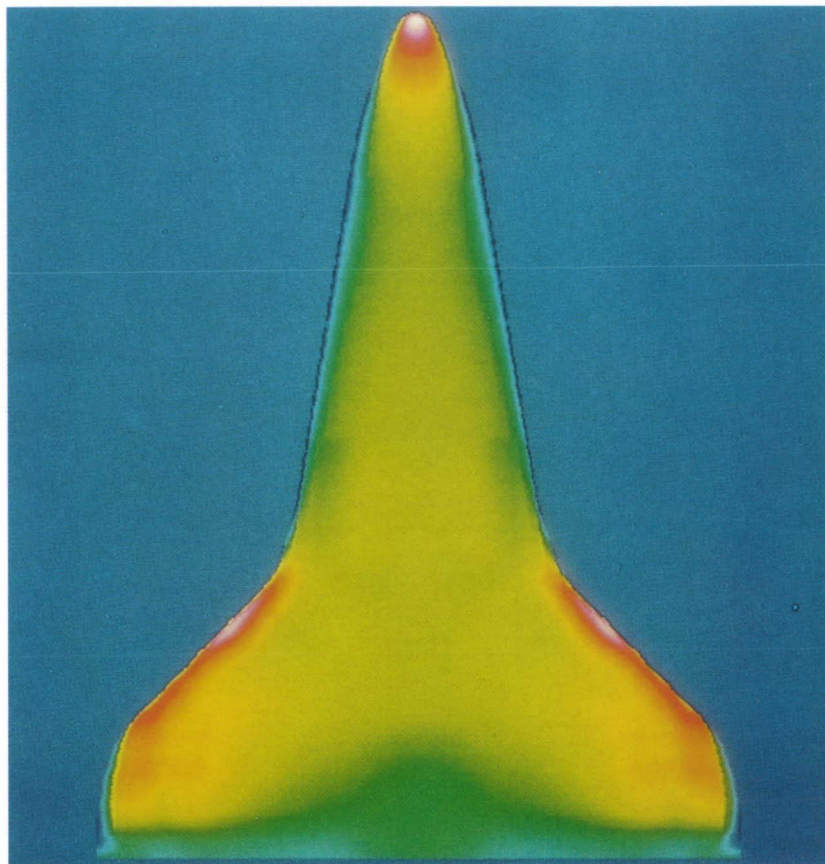
formed 3-D CFD calculations over the last 20 years. However, with its acquisition of the VPS-32 computer in 1984, Langley became the first aerospace research laboratory to have the capability to do routine 3-D calculations with advanced CFD models. Its research efforts are now complemented by remote access to the Numerical Aerodynamic Simulator (NAS) supercomputer at NASA Ames.

Langley's in-house and out-of-house funded work in CFD has resulted in a variety of computer programs or codes that have found widespread use in industry. In the subsonic regime, a wing computationally designed at Langley to minimize drag has been flight tested on the Cessna 210 aircraft and was found to result in a speed increase of 12 knots over its original design. In addition, the aircraft was much less likely to undergo an abrupt stall. At transonic speeds, Langley-funded and developed codes were used by Grumman in the design of its X-29 Forward Swept Wing Demonstration Aircraft, which may herald the look of future military fighters. While verification of the older non-computationally designed F-16 required 12,000 wind tunnel hours from 1971 to 1982, the use of Langley's CFD codes resulted in a require-
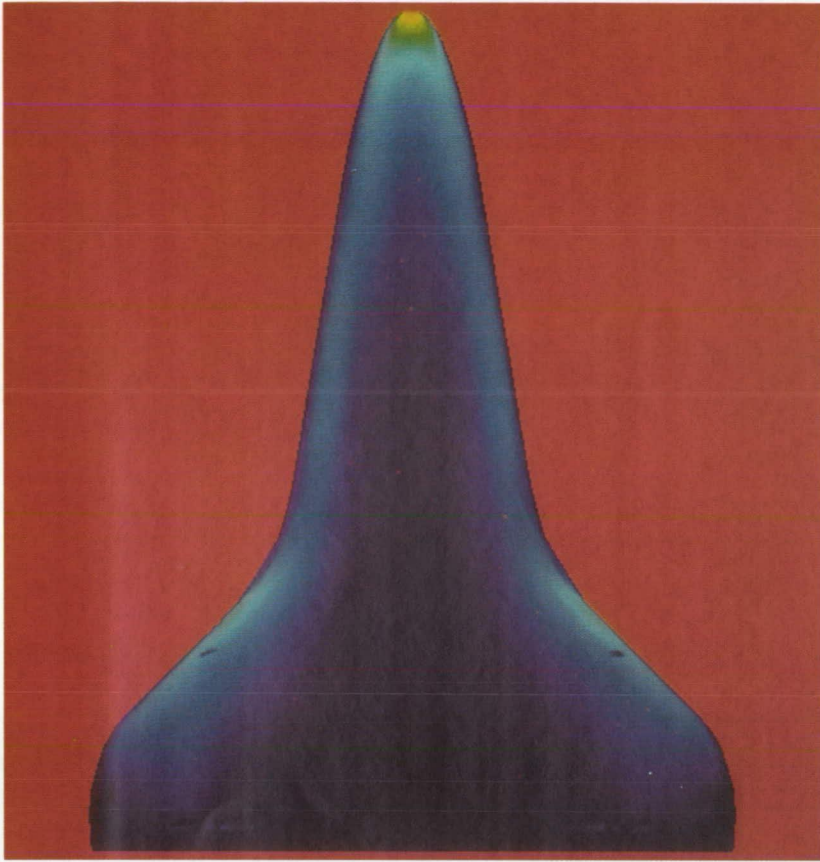
Computational methods will soon enable aerospace engineers to investigate such detailed phenomena as the flow of hot gases between parts of a vehicle. Below, Langley scientists calculate the flow of hot gases (blue cool, white hot) into the gap between the Shuttle Orbiter's wing and elevon to determine whether critical seals can protect uninsulated interior parts of the Shuttle's wing.
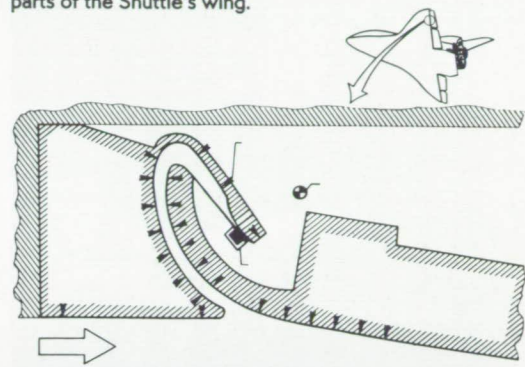

ment of only 160 hours for transonic and supersonic wind tunnel testing for the X-29. At supersonic and hypersonic speeds, the Langley-developed HALIS/ AA3DBL codes are currently being used to analyze shuttle re-entry data, and can be applied in the future to help design Shuttle II and to help determine the aerodynamic characteristics of the "SWINT/QUICK" hypersonic missiles.

Other CFD codes are used extensively throughout the world. Both the Gulfstream III & IV and the Boeing 757 wings were computationally designed. Codes were used extensively for the Grumman HiMAT, the NAVY/ Grumman A-6 attack bomber, and an advanced Cessna propeller. Code use saved an estimated $4 million on the cost of designing a Gulfstream swept wing executive transport. McDonnell employed codes in the design of the F/ A-18 Hornet and the AV-8B Harrier 2. Boeing Commercial Aircraft estimates that approximately 80% of a transport wing's final cruise-optimized design is now arrived at by use of CFD methods. And, by using a computational code, Rockwell International has recorded a 30% improvement in the ratio of lift to drag for the B-70.
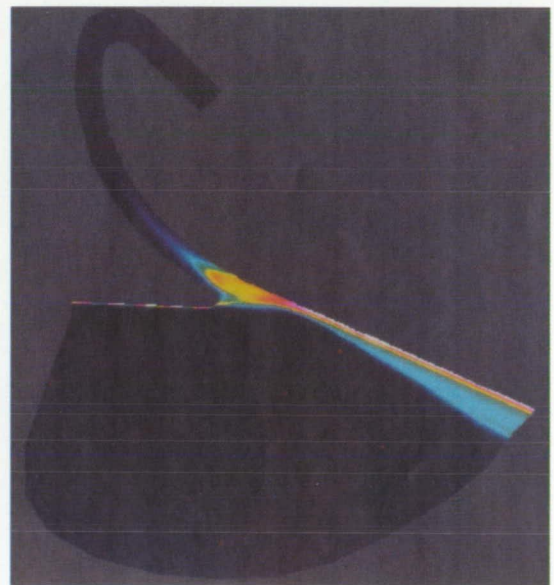
▪SUPERCOMPUTERS
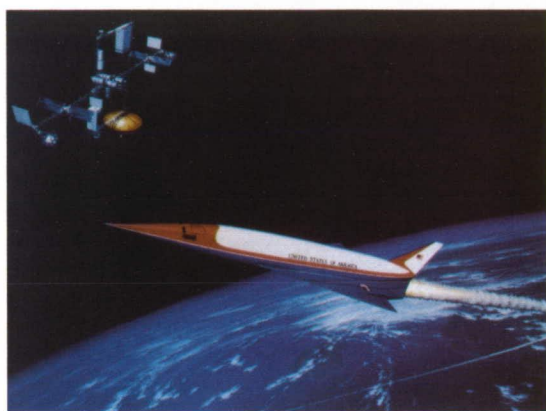
The VPS-32 Supercomputer is helping Langley researchers to design the next generation of hypersonic vehicles, such as the Aero-Space Plane (above).

Throughout its brief history, CFD developments have pushed computer resources to the limit. Improvements in both computer speed and computer architecture have made many CFD advances possible, yet they have also presented new challenges. The need for more detailed physics has forced Langley researchers to develop more efficient codes to take advantage of new computer capabilities.

The serial "one-operation-at-a-time" mode of computers led to the so-called "von Neumann bottleneck," which inhibited computer performance. As a result, computer architecture is beginning to play a significant role. "Vectorizing" a supercomputer, or configuring it so that it can perform identical arithmetic operations on an array of numbers, saves both time and memory capacity.
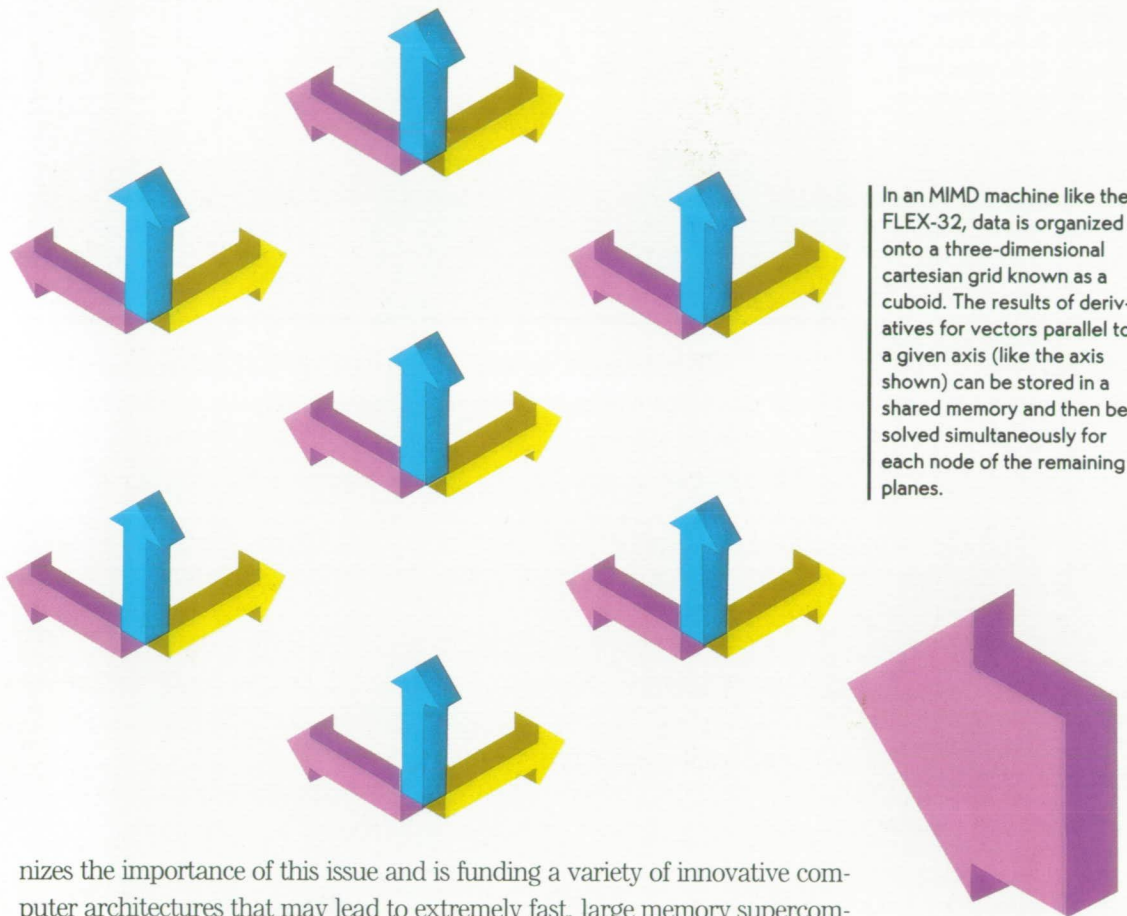
In order to circumvent the limits posed by component performance, computer designers have resorted to engaging more than one functional unit at a time. This approach, termed parallel processing, has the potential for very significant speed improvements.

True parallel processing is the direction the computer industry is headed. Machines of the multiple instruction stream/multiple data stream (MIMD) type, which incorporate a number of full functional central processing units (CPUs), are currently undergoing intensive development. In a MIMD computer, each CPU can execute a separate program and work independently or in concert with other CPUs in the machine.

One issue of paramount importance is how these high performance MIMD machines will be brought to bear productively on scientific problems. With vector computers, those algorithms which use the vector arithmetic units most effectively often require more total arithmetic calculations to achieve an answer than algorithms which vectorize less effectively. Researchers are studying the tradeoffs between computational efficiency (the speed of algorithms) and algorithm efficiency (the number of operations required to achieve a solution) to establish the algorithm with the highest net efficiency for a given problem.

CFD is moving into a period when research into the algorithm/computer architecture relationship is crucial. Langley Research Center recog-

In an MIMD machine like the FLEX-32, data is organized onto a three-dimensional cartesian grid known as a cuboid. The results of derivatives for vectors parallel to a given axis (like the axis shown) can be stored in a shared memory and then be solved simultaneously for each node of the remaining planes.

nizes the importance of this issue and is funding a variety of innovative computer architectures that may lead to extremely fast, large memory supercomputers whose designs are optimized for solving Navier-Stokes or Euler equations.

To gain experience with parallel processing, Langley acquired a low-cost microprocessor-based MIMD machine with an architecture as similar as possible to the next generation of supercomputers. This FLEX/32 Multicomputer from the Flexible Computer Corporation has 20 independent processors and a common bus from which run ten local buses. On each local bus are two 32-bit microprocessor-based single-board computers (CIC), each with a hardware floating point accelerator, a memory management unit and one megabyte of memory. The complete system has 30 megabytes of memory, or 7.6 million 32-bit words of memory. This memory can be partitioned either as shared by all processors or as local to individual processors, providing even greater flexibility.

Located at NASA Ames Research Center, the NAS Cray 2 Supercomputer is one of the world's most advanced computers. Here, Langley researchers are using the NAS to develop codes that model compressible flows at high Mach numbers. By using existing knowledge of the behavior of simpler incompressible fluid flow, scientists can study the similarities that exist in the behavior of compressible flows.
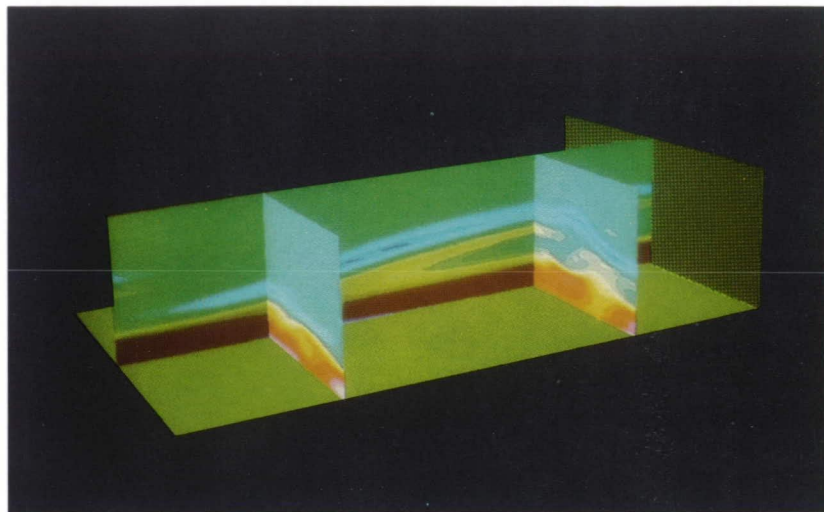
To implement parallel code on the FLEX/32, extensions of FORTRAN and C have been developed to process and facilitate data sharing. These extended languages are called Concurrent FORTRAN and Concurrent C. With these languages, the FLEX/32 is already being used in pioneering studies of parallel algorithm implementation in both CFD and Computational Structural Mechanics.

Jointly developed by scientists at Princeton University and Langley Research Center, the Princeton/Langley Navier-Stokes computer will have 128 "nodes" in the configuration shown here. Each "node" will be capable of performing one-half billion floating point calculations per second. This unique architecture may facilitate solutions for complete Navier-Stokes equations that employ 60 derivatives.
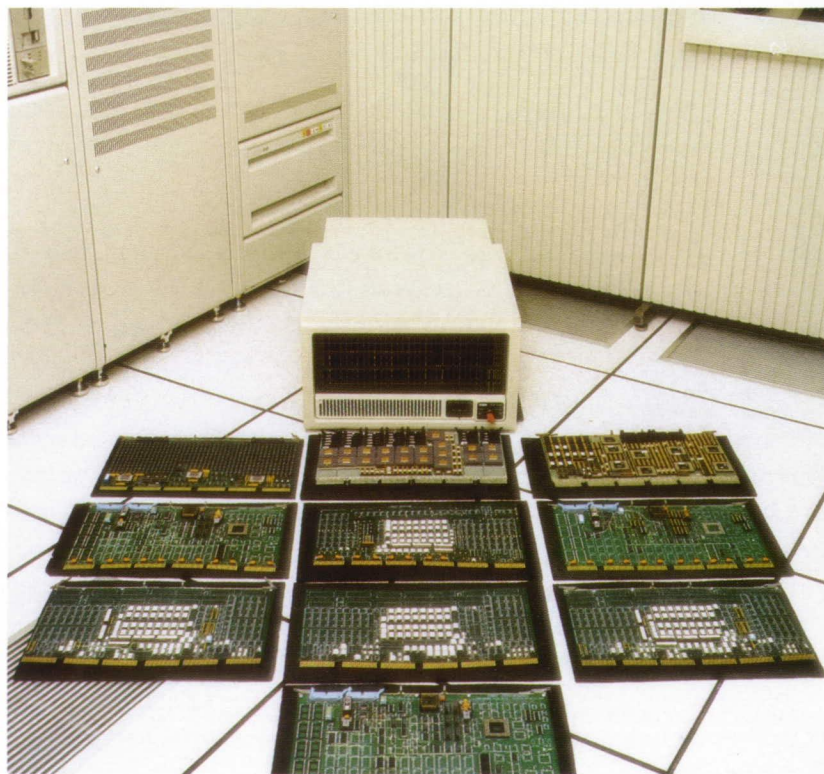
A three-dimensional, spectral, time-dependent, compressible Navier-Stokes code was written to study the transition of flat plate boundary-layer flows from a laminar to a turbulent state. As the flow evolves toward a smaller scale, the number of required grid points to resolve its structure increases, resulting in increased memory usage and a substantially larger amount of computer time. To reduce the execution time, the code was adapted to run on the FLEX/32.

With this adapted Navier-Stokes code, the bulk of the processor time is spent evaluating the necessary derivatives of velocity, pressure and density in all three coordinate directions. An efficient matrix multiplication algorithm, capable of running on an arbitrary number of processors, enables researchers to evaluate derivatives more readily; and a full parallel Navier-Stokes code can be implemented. The code is duplicated in each processor, which handles exactly 1/NP of the operation count. The FLEX/32's shared memory can accommodate a full Navier-Stokes calculation on a 32x32x32 grid. Preliminary results show that, with a 16-processor configuration, the execution of a derivative calculation operates 13 times faster than a single processor if the matrix is of the dimension 128x128x128.

Langley is also exploring CFD via parallel computing through its support of the Navier-Stokes Computer (NSC) project under the direction of Professors Nosenchuck and Littman at Princeton University. Close interactions between Langley and Princeton researchers are leading to a high performance computer specially tuned to the requirement of algorithms for the Navier-Stokes equations. The NSC will then be able to produce extremely rapid, high resolution solutions to the time-dependent Navier-Stokes equations. In short, it will be a computer with two levels of parallelism – a fine-grained parallelism achieved through a novel design of individual nodes. On the coarse-grained level, researchers envision on the order of 128 nodes connected through a hypercube network. The Concurrent Computer Corporation of Tinton Falls, New Jersey, is developing NSC technology further into a commercial supercomputer.

The key architectural features of an individual node are: 16 separate memory planes which feed the arithmetic logic unit (ALU) simultaneously; an ALU consisting of 24 vector pipelines arrangeable in a reconfigurable tree structure; and a non-blocking switch which rearranges data coming from the memory to the ALU and also the data going back to the memory from the ALU. Each node is capable of a peak speed of 480 Mflops and has a memory of 256 million words.

Langley researchers are currently testing the details of this design and are investigating the best methods for implementing CFD algorithms on this supercomputer.

ALGORITHM RESEARCH

Imagine an air cell the size of an ice cube. What forces affect and control that cell's motion and its interaction with an airplane wing? When that cell slides along the top or bottom wing surface, pressure changes occur, ultimately defining the lift and drag of the wing. At the same time, the cell is affected by the other air cells that surround it.

By using CFD to solve a set of partial differential equations (PDEs), aerospace CFD researchers are seeking to define the forces that act upon our air cell as it passes over an aircraft surface. Newton's well known $(F = MA)$ law enables a CFD researcher to calculate the aircell's acceleration and velocity as it moves along the wing, deriving a straightforward differential equation of the resultant force (F) acting on that cell. This "Euler equation" is then used in regimes where the frictional (also called shear or viscous) forces acting on the aircell are small enough to be ignored. When researchers include those frictional forces that act on each of the aircell's six sides, they use an extended set of equations called the Navier-Stokes equations.

First published in 1823, the full set of Navier-Stokes equations are classified as nonlinear, second-order partial differential equations. When expressed in three dimensions, these equations include 60 partial-derivative terms, making them computationally too difficult to solve with enough precision for the practical design of aircraft.

However, CFD researchers are using three approximations in the absence of exact Navier-Stokes solutions. The first, called "linearized inviscid," has become widely used in industry because of its simplicity (rather than 60 terms, the N-S equation is abbreviated to only three) and its surprising accuracy. Here, aircraft geometries are modeled by using a large number of form-fitting surface panels, and the flow properties above each panel are then calculated in turn. This "panel method" has been employed since 1930 on such problems as subsonic and supersonic pressure loads, lift and side forces, and even induced vortex drag. More recently, it has been used successfully to solve 3-D aerodynamic flow problems of idealized, but complete aircraft and spacecraft.
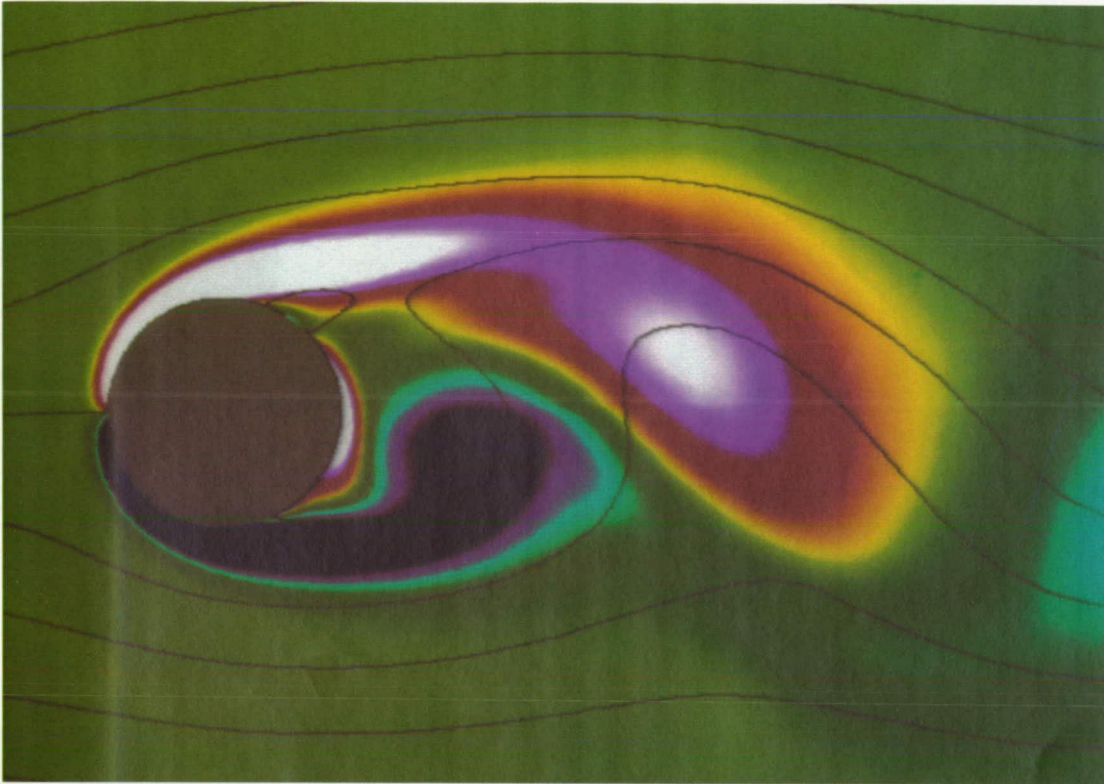
Langley researchers developed the computational fluid code CFL2D to predict two-dimensional unsteady flows. These two computer-generated images display vortices and flow separation for a cylinder and a wing beyond stall (clockwise flows are white to red; counterclockwise flows, black to blue). In both cases, researchers can see vortices being formed and shed in a periodic manner.

15
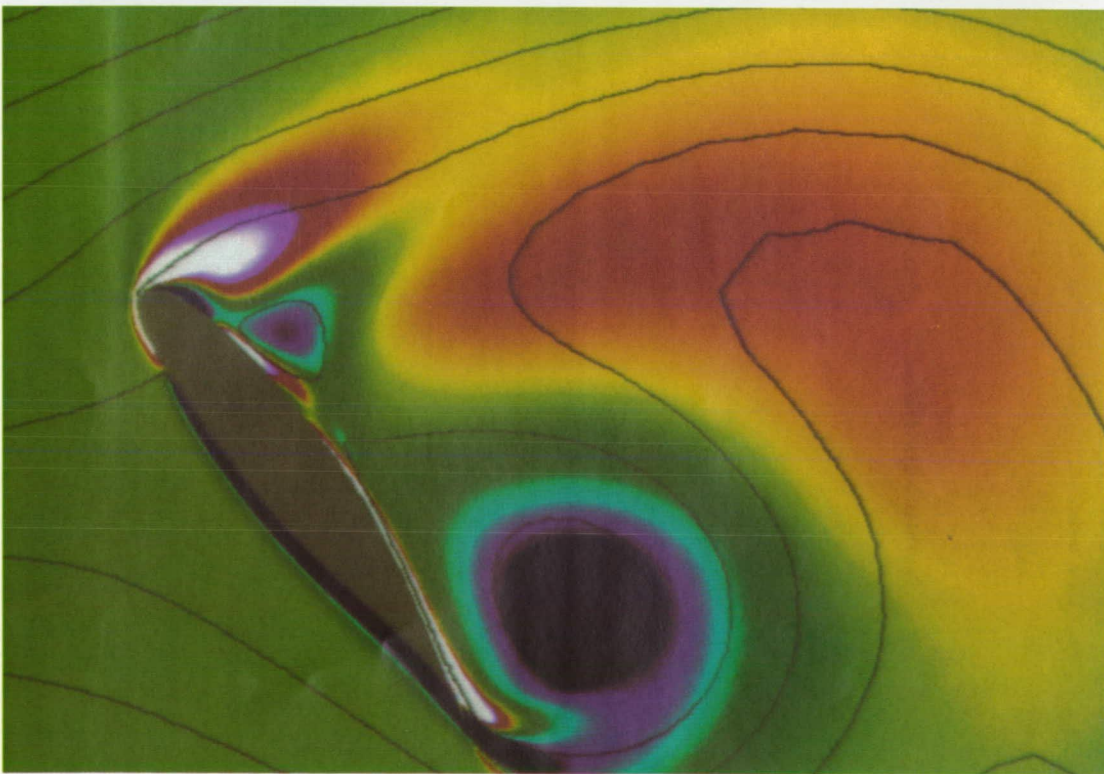
Using a reduced set of the
Navier-Stokes equations
coupled with a numerical
eddy-viscosity model for
turbulent flow, these com-
puter simulations show the
flow of particles over a
transport-type transonic
wing at M = 0.84. The first
example (left) shows the
simulated flow reversing
near the outboard section
of the wing. In the second
example (right), particle
traces show the movement
of air from the high pressure
lower side of the wing to
the low pressure upper sur-
face, resulting in a wingtip
vortex.
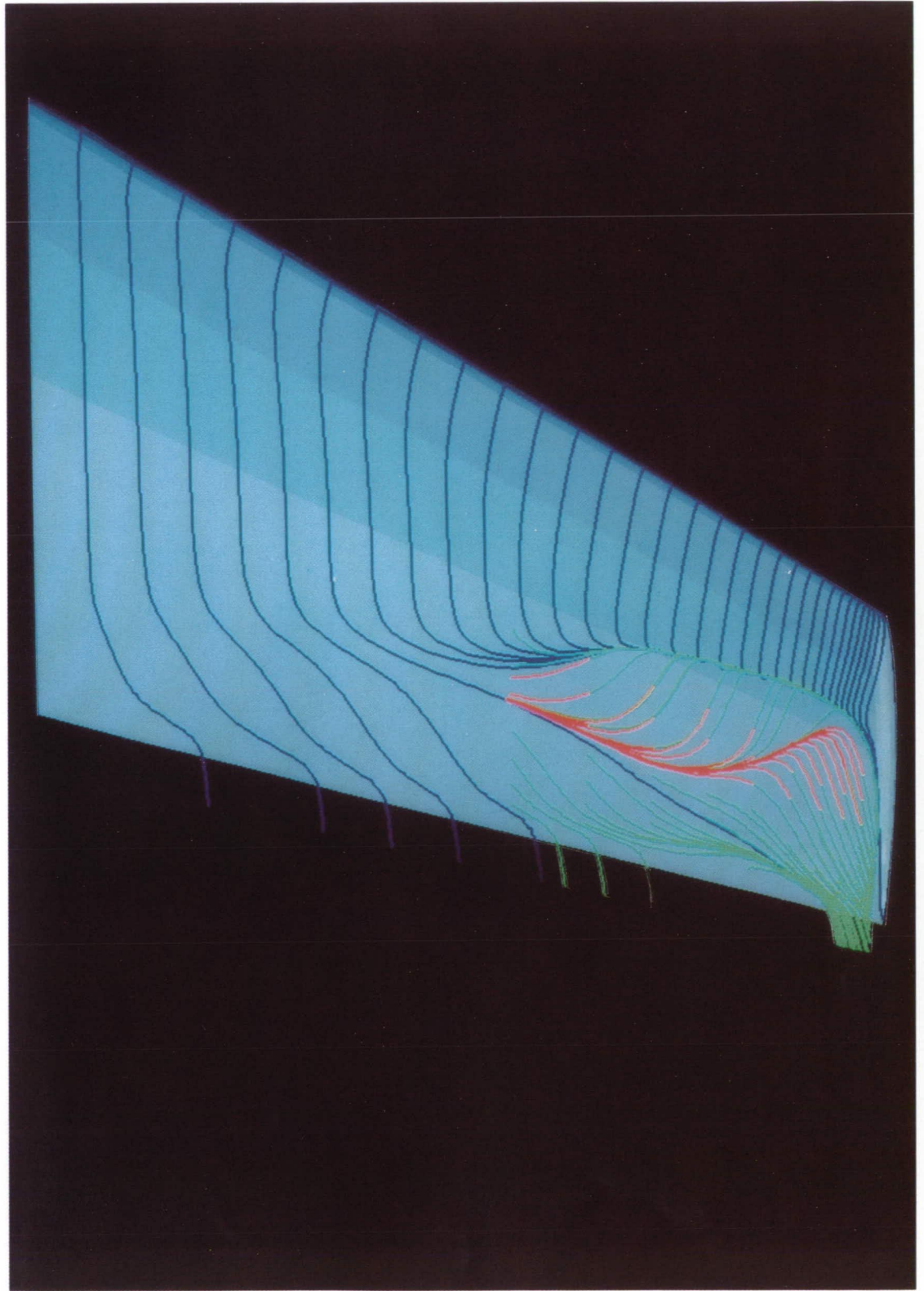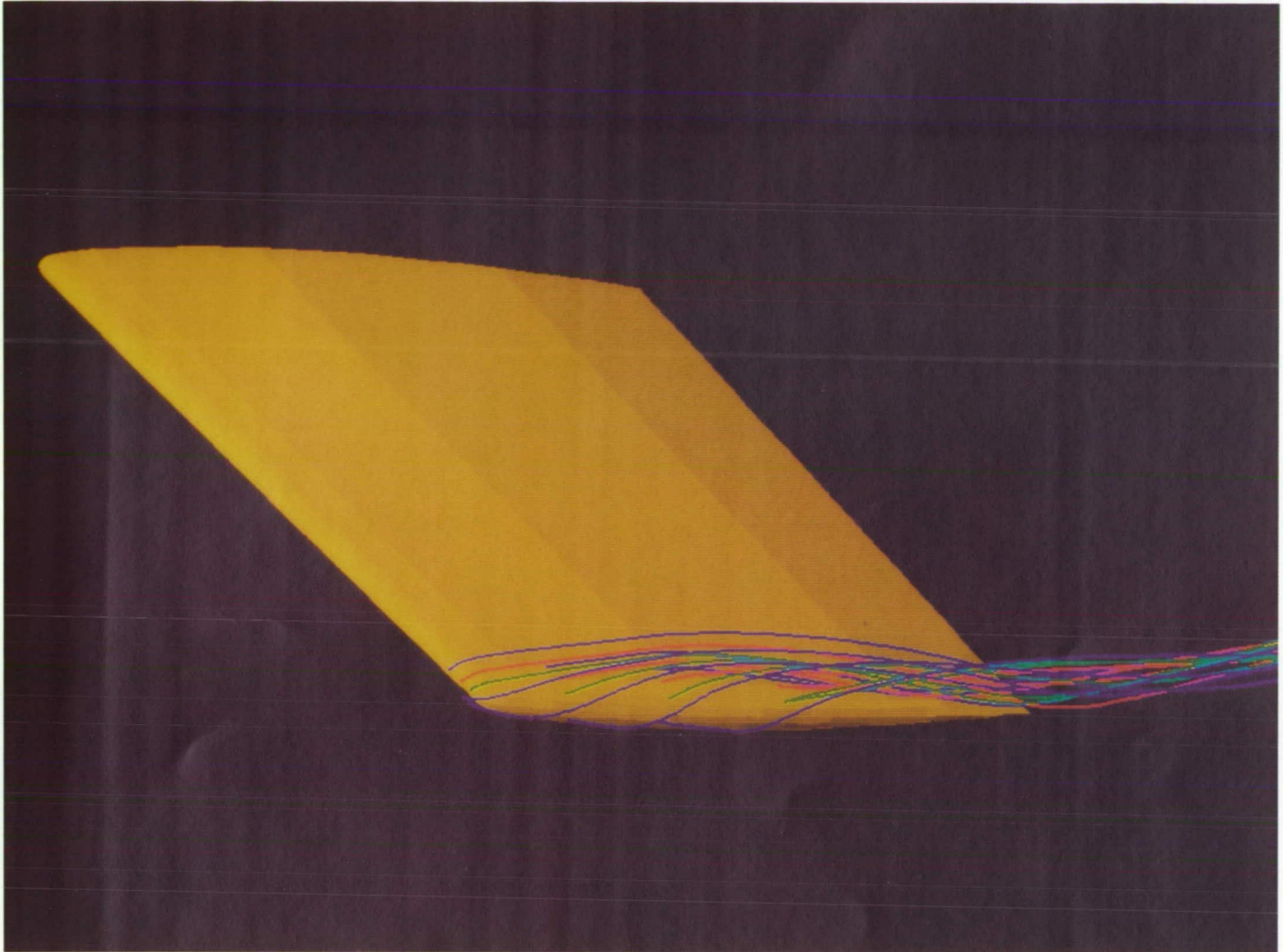
The next, more sophisticated equation, called "nonlinear invis-
cid," has received the most attention from CFD researchers at Langley
Research Center, as well as other NASA centers, industry and academia. This
method results in 27 partial-derivative terms, as opposed to the original 60 in
N-S equations. Only the viscous effects are not included, although viscous
corrections can be added. An added complexity of this nonlinear approxima-
tion is the requirement to solve the equations throughout space, not only on the
vehicle surface, as in the linearized approximation. Because of its computa-
tional complexity, the nonlinear inviscid method was rarely employed before
computers appeared. However, NASA CFD researchers now employ this
method not only to compute the same variables as the linearized inviscid
method, but also to compute transonic pressure loads and a variety of situa-
tions involving shock waves.

The nonlinear inviscid method has enabled Langley researchers to
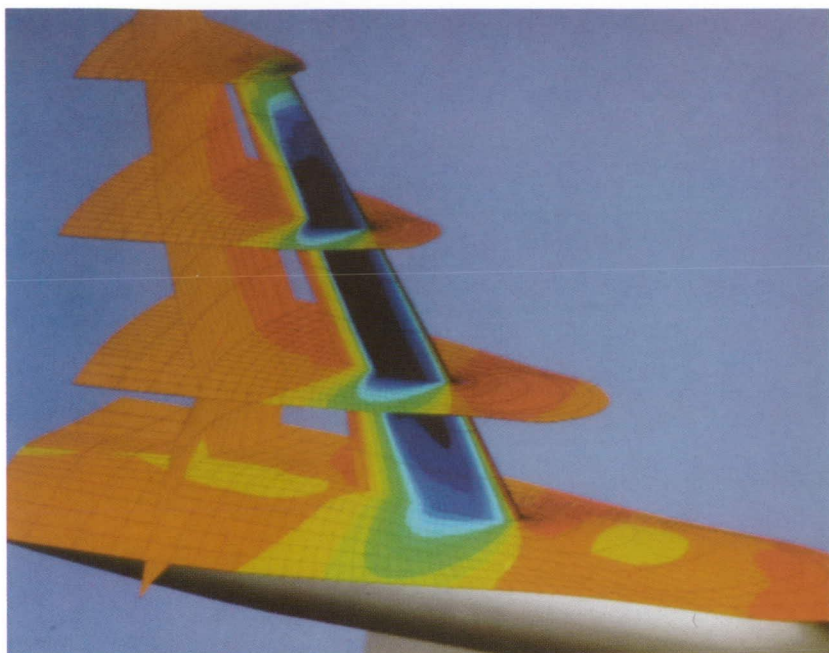
Multigrid approaches to computational fluid flows promise significant improvements in accuracy and speed. The lift history in this three-dimensional flow field was calculated with multigrid in less than 40 iterations; the conventional single grid method requires more than 400 iterations.

develop codes that are widely used in industry, with successful applications in the HIMAT aircraft and Space Shuttle designs, as well as solutions to internal flow problems involving inlets and nozzles. Langley researchers can also couple equations that describe a flow field with equations that define the structural behavior of an aircraft to study the aeroelastic deformations and loadings induced by the flow field. Similarly, the coupling of CFD with the aircraft structural dynamics is used to study the aeroelastic phenomena of flutter, buffet and gust response.

The solution of a flow involves three steps. First, the problem is modeled by a set of partial differential equations (PDE's) and boundary conditions. This model expresses the conservation laws of mass, momentum and energy. Then, the scientist selects either the linearized or nonlinear inviscid model, or the viscous Navier-Stokes equations.

Second, the flow domain is divided into many tiny subdomains represented by cells or points. The PDEs are approximated by a set of discrete algebraic relations for each cell. The algebraic relations derive from finite difference, finite volume, finite element or spectral approximations of the governing equations. Insight into the physics of a problem guides the successful formulation of an algebraic approximation of the PDEs, as in the upwind differencing techniques which attempt to maintain the influence of various waves traveling from cell to cell. The number of cells needed to describe a solution accurately is governed by the complexity of the problem. For example, with a Boeing 747 flying at $M = 0.84$, there may be five hundred thousand to several million cells necessary for an accurate resolution of all inviscid flow phenomena. With five or more equations to be solved at each cell, the total number of coupled simultaneous equations needed to simulate a flow dictates the need for highly efficient algorithms.
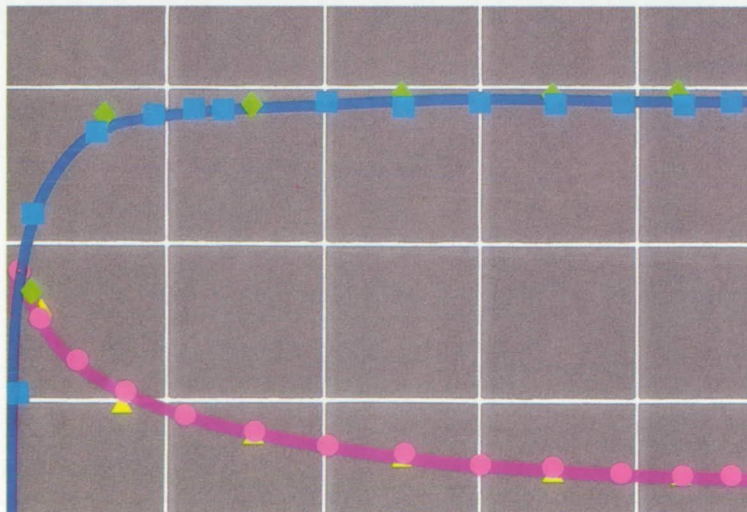
The third and final step for flow simulation occurs when the discrete equations are solved with a procedure that effectively utilizes both the computer architecture and a solution algorithm appropriate for the class of equations being solved.

A solution algorithm can be thought of as a way of solving a large matrix equation: $\underline{A}dq = r$. The vector r defines the physics of the problem. The vector dq represents an increment in the dependent variables (mass, momentum and energy). The matrix $\underline{A}$ represents a linearization of the algebraic expressions defining the physics in the vector residual r. The simplest algorithms result when the matrix $\underline{A}$ is diagonal.
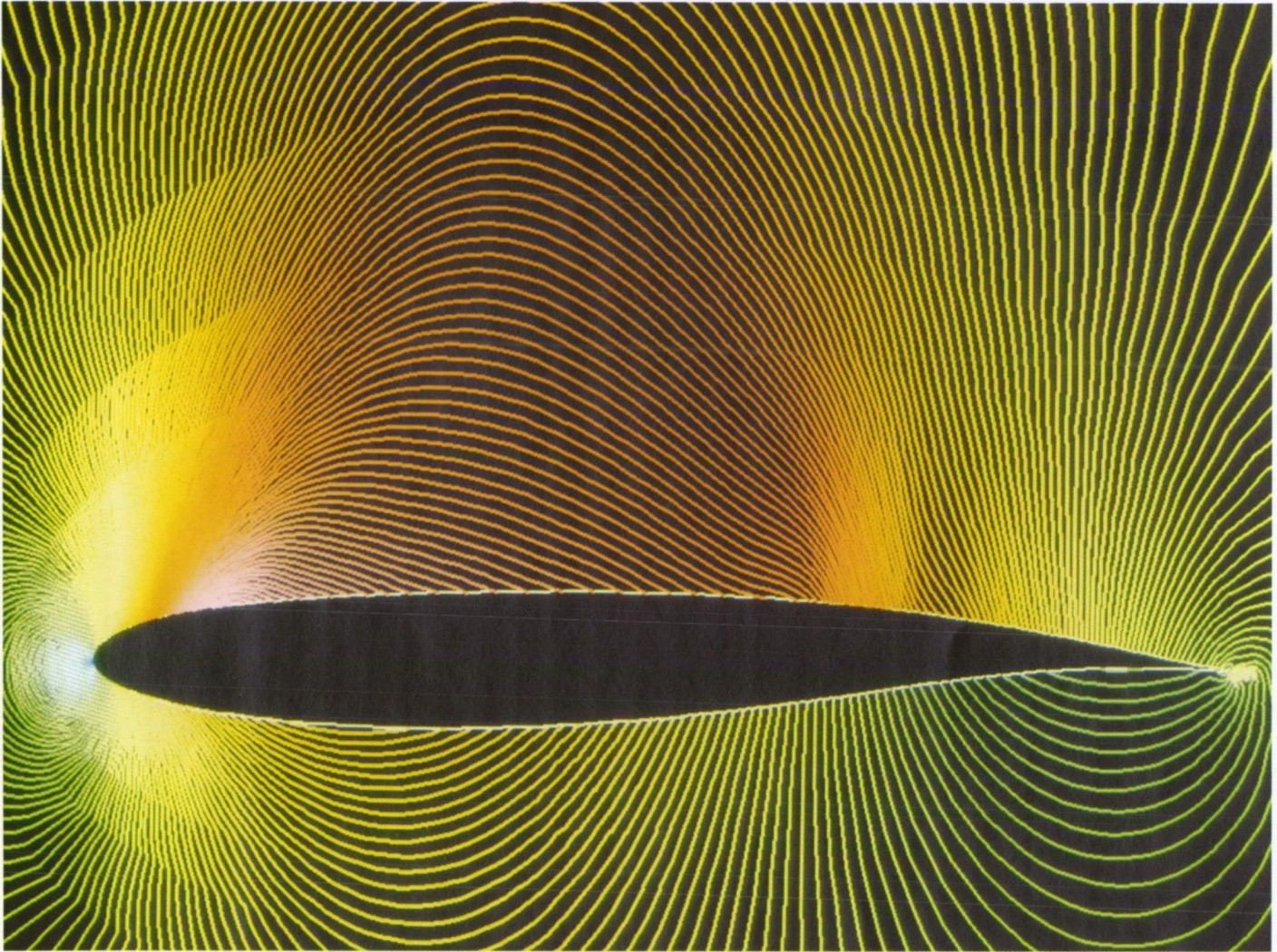
Much of algorithm research focuses on developing more efficient CFD codes. The graph (right) shows spectral approximations of a chemically-reacting flow field. The solid curve represents the slower finite difference methods of calculation, using a 101 point grid to calculate values of the reaction; red data points show a spectral approximation that uses only 17 points; and green shows an approximation for 9 points. More efficient CFD codes can speed the design process.
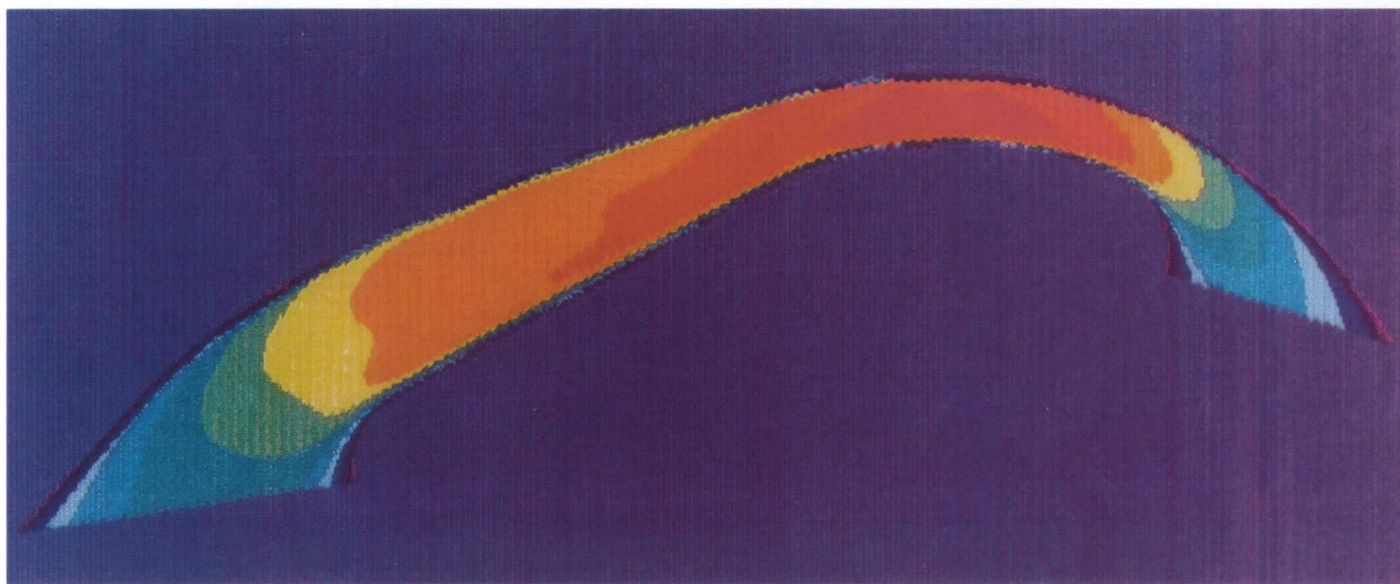
A method called spectral multigrid combines spectral approximations with high efficiency multigrid techniques, reducing computer time and computer memory. Spectral multigrid is providing accurate solutions to complex transonic potential flows. The solution (above) was generated for an airfoil at $M = 0.75$, using a grid of only 80×29 points.

This means that the algorithm is explicit, and changes in the dependent variables dq are explicit functions of a single equation at each cell. Such techniques are easy to program, but they may require many iterations to drive the residual vector r to zero because of stability constraints. In implicit methods, where a change in a dependent variable is implicitly a function of several equations at neighboring cells, stability constraints are relaxed; however, the solution procedure is considerably more involved. In order to simplify the solution process, approximate factorization techniques are most often utilized in implicit methods.
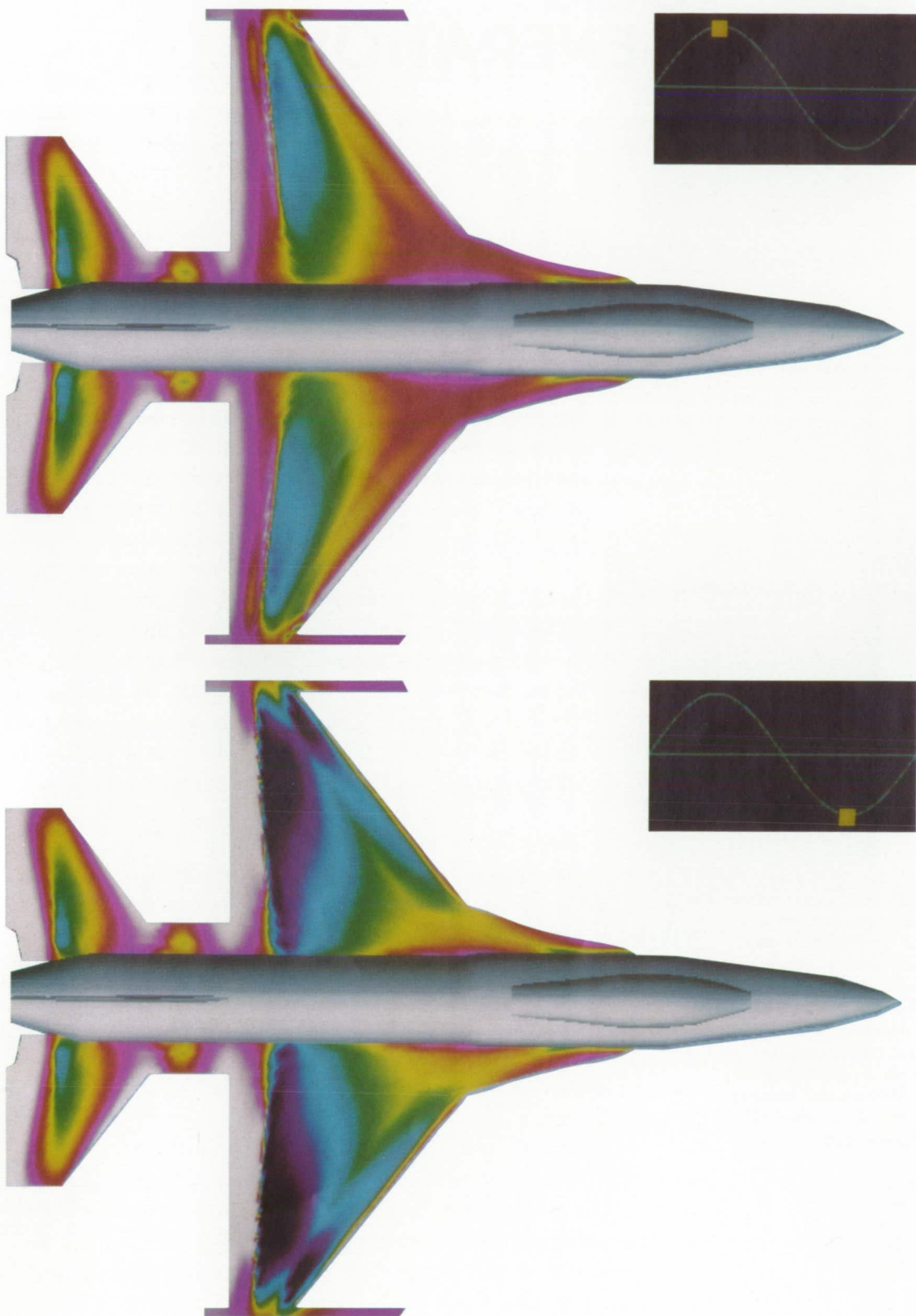
Acceleration techniques can enhance the convergence of an algorithm. One such technique pioneered at the Langley Research Center is called multigrid. With multigrid, the same problem is solved simultaneously on a set of successively coarser grids. Information from the coarser grids enables researchers to accelerate the convergence of the finest grid. Multigrid techniques promise iterative techniques which converge to a given accuracy on a fixed number of iterations, regardless of the grid size. Thus, the use of multigrid has enabled researchers to achieve one order of magnitude improvement in efficiency.

Research on new algorithms also includes taking advantage of computers with vector processors and massively parallel processor architectures for solving systems of equations. Algorithms that are very fast on one architecture may be extremely slow on another. Therefore, some of the greatest potential in advancing state-of-the-art computational fluid dynamics exists in this area. For example, Langley has shown speedups of over one order of magnitude in studies of boundary-layer transition using the parallel architecture FLEX/32 computer.
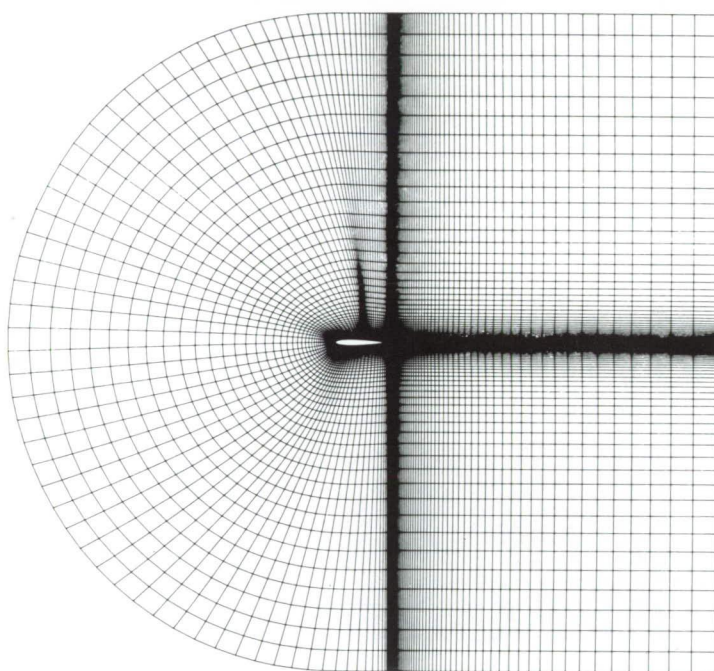
The computational fluid code LAURA has been developed to meet such special problems of hypersonic flow fields as high temperatures and strong discontinuities. The code uses finite volume approximations to maintain accurate results for mass, momentum and energy. The temperature contours shown here are for a proposed aerobrake; the temperature ratios across the bow shock of such a vehicle at M = 32 approach a factor of 200.

A recent NASA Langley development, the CAP-TSD code, enables designers to analyze and predict aerodynamic interference and flutter characteristics on complete aircraft. These images show pressure contours for an F-16 aircraft pitching during flight at $M = 0.9$. The upper image shows the aircraft at its maximum pitch angle, while the lower image shows it at its minimum (flow compression is yellow through white; flow expansion, green through black).
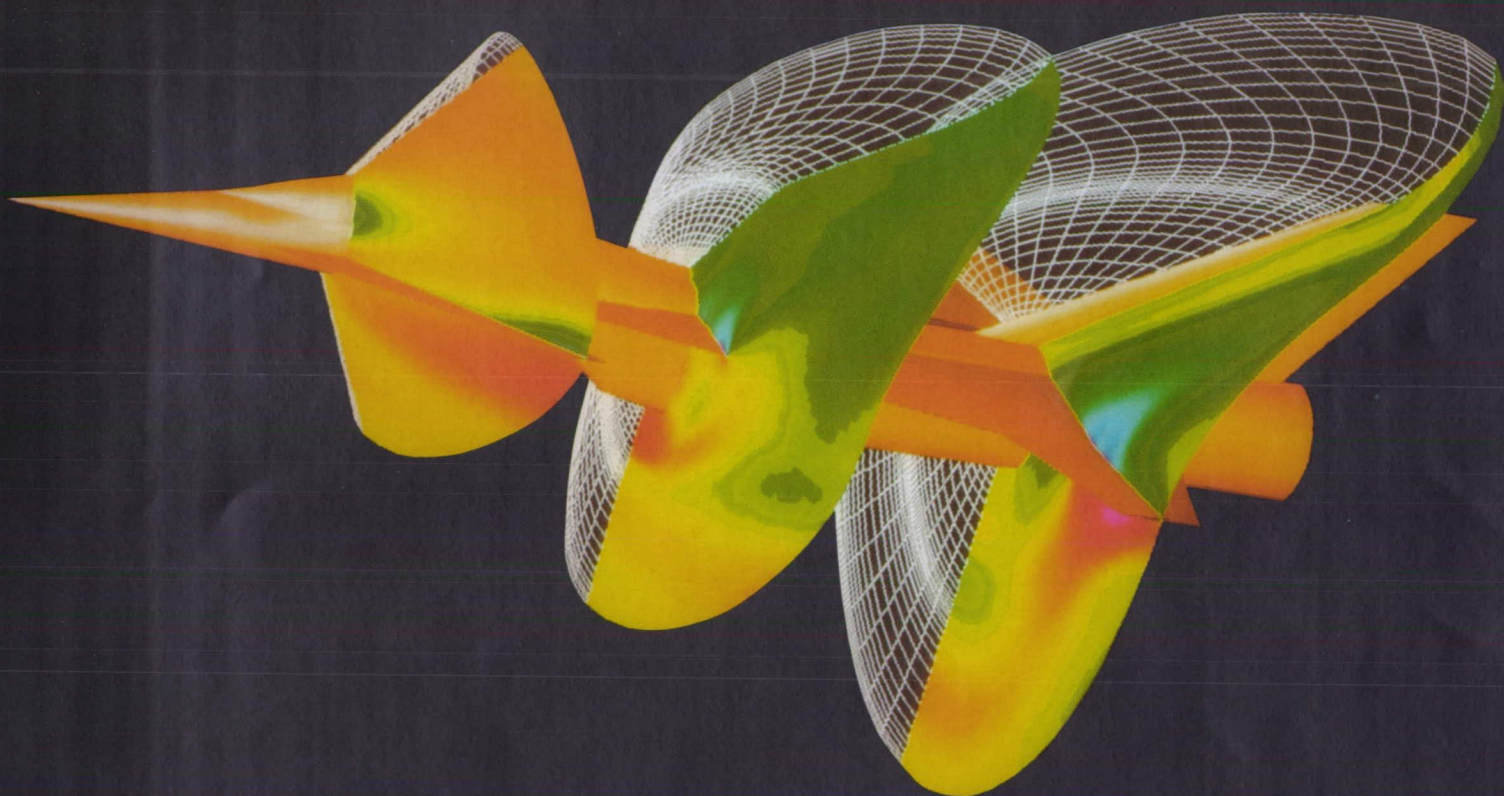
GRID GENERATION

Although the calculation of airflow about aerodynamic models is now mathematically tractable, it still presents enormous computational difficulties. If the aircells are too large, the accuracy of the CFD solution will most likely be inadequate. Thus, CFD researchers employ many aircells that are extremely small compared to the aerodynamic model, and they vary the size of these air cells. The general strategy is to vary the size of cells and place more and smaller cells where the most activity occurs.

The aircells are defined by points, and the total collection of points along with their relations to neighboring points is called a "grid" or "mesh." Each grid point has from five to thirty numerical quantities associated with it, and these quantities are related to each other and to the set of quantities at adjoining grid points. Even a relatively simple calculation to simulate turbulent airflow over a wing requires about one million grid points. A complete calculation can require 10^{13} computations or more.

Constructing computational grids has become an important aspect of CFD. In fact, in the late 1970s, the computation of grids, "grid generation," was considered to be a pacing item for CFD. In 1980 Langley held the first grid generation workshop to be attended by both national and international CFD researchers. This meeting was a milestone for the development of grid generation technology. Grid generation has since become so important to CFD that it is a research topic unto itself. Today, researchers can construct grids for highly-complex three-dimensional flow field problems; in the future, developments in grid generation should keep pace with computer technology and algorithm development for CFD applications.

There are two characteristics that are desirable when grids are generated. They are: (1) grid points should conform to the boundaries of objects about or within which a CFD solution will be computed, and (2) grid points should be concentrated where solution gradients are high. For example, in a viscous solution, grid lines must be "packed in" near solid boundaries to resolve the large velocity gradient that occurs in boundary layers.

An algebraic method of grid generation (above) called the "two-boundary" technique has many desirable features. It conforms to the shape of the boundaries, is as orthogonal as possible, and "fits" the physics of a problem because it finely spaces the grid where high gradients exist.

Developing a grid by direct algebraic construction can dramatically simplify programming and lead to reduced development time. Once the grid has been completed in three dimensions (right), Langley researchers can transform fluid dynamic equations to fit this new curvilinear coordinate system.

Grid Generation

Generating grids about complex geometries such as complete aircraft can be extremely difficult. Dual block grids solve the governing equations for separate blocks or zones. The CFD code then provides an interface for solutions developed in each zone. The lower figure shows a dual block grid for a fighter aircraft; the top figure displays pressure coefficients generated at M = 2.0.
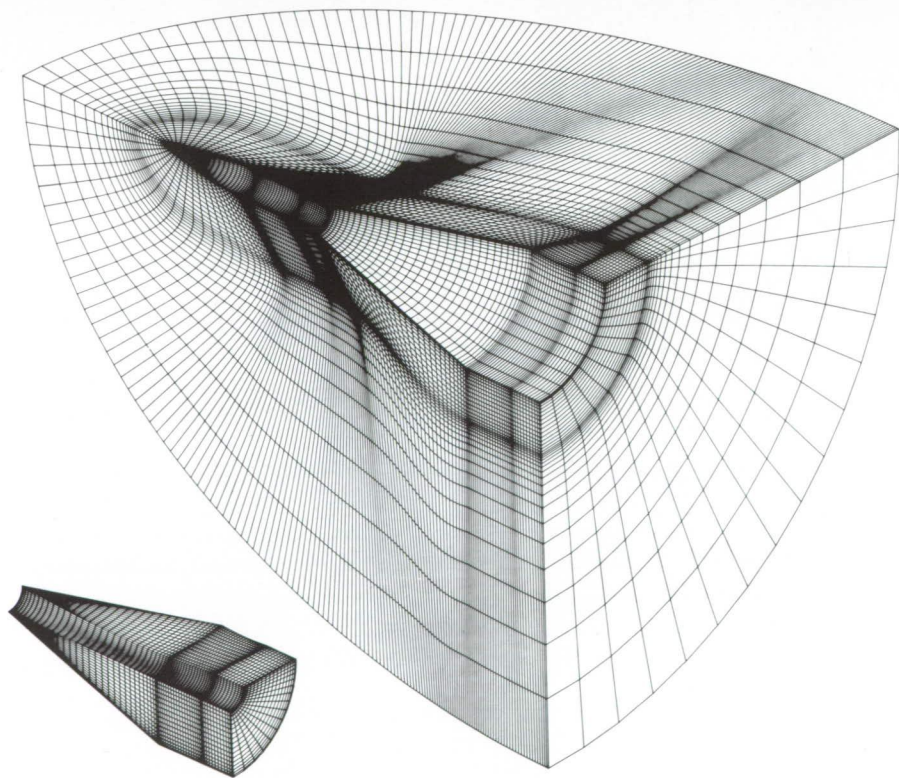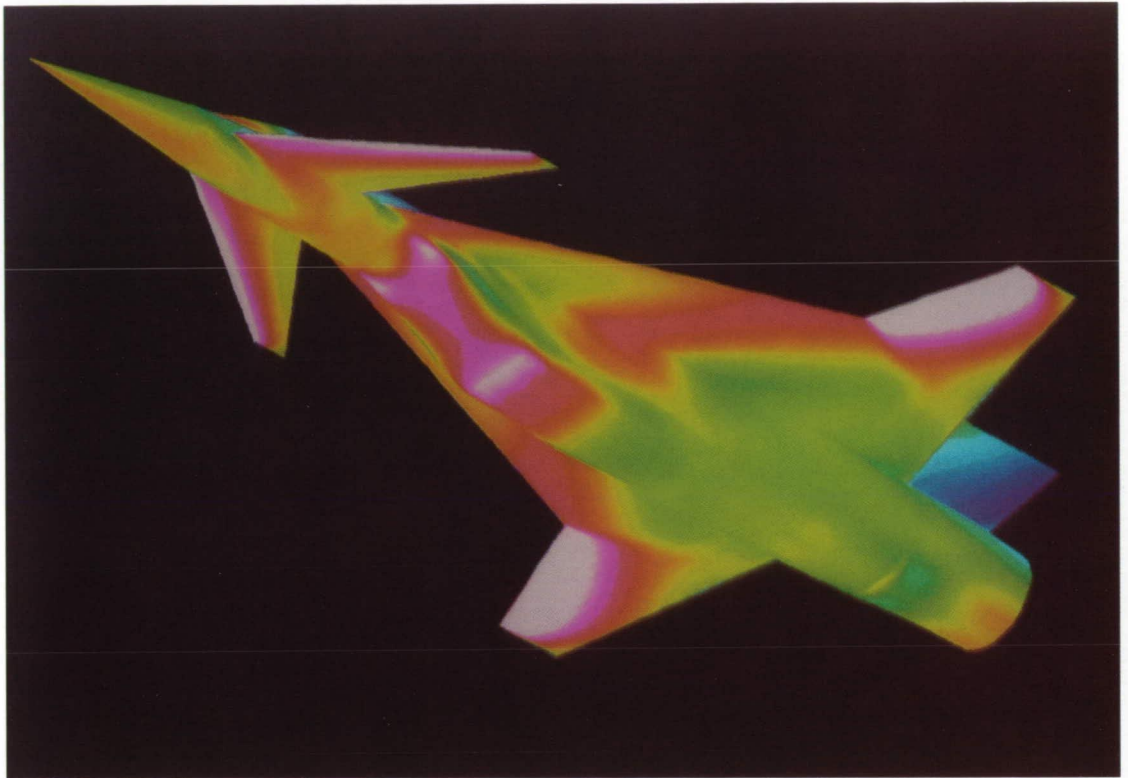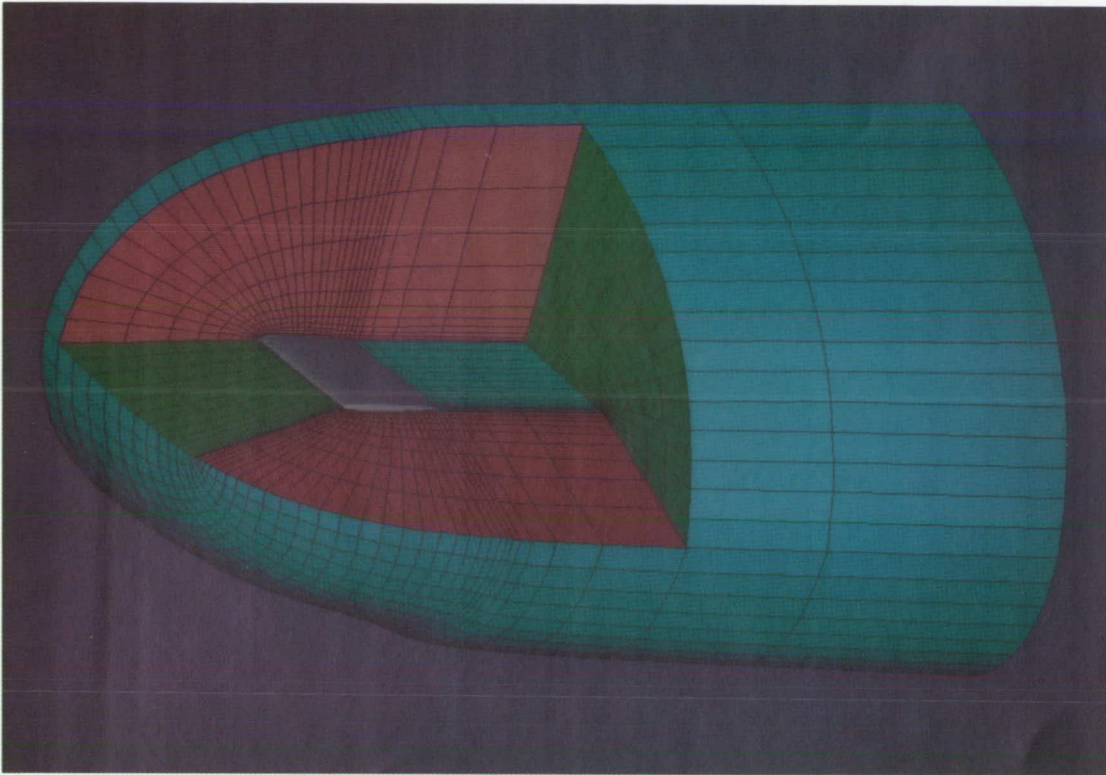
This partial view shows a three-dimensional grid generated from a surface patch description of an aircraft wing (blue). Grid points have been tightly spaced in the wake region of the wing, enabling researchers to study such particular phenomena as viscous flow.

Grids may be placed in two broad categories: structured and unstructured. These categories may be further divided into two types: stationary and adaptive. Stationary grids are those that are generated in advance of a flow field computation and never change thereafter. Adaptive grids change throughout the course of a CFD calculation. For example, the grid points may move to regions of high gradients in the flow field as the solution progresses.

The most prevalent grids in use today are structured grids. These have a one-to-one correspondence to some general curvilinear coordinate system. Once the grid is generated, the fluid dynamics equations are transformed to the curvilinear frame of reference using techniques from tensor analysis. Then, the appropriate derivative approximations are made, and the CFD problem is solved in the new "computational space." A nice feature of structured methodologies is that the "computational space" can be rectangular with unit grid point spacing, regardless of the shape or the grid point spacing in the physical space.

Two approaches have been developed to generate structured curvilinear grids. They are categorized as differential and algebraic. Differential methods require the solution of a system of partial differential equations, and algebraic methods require the creation of specific formulas.

The dominant differential method is the elliptic method pioneered at Mississippi State University in the early 1970s under Langley sponsorship.
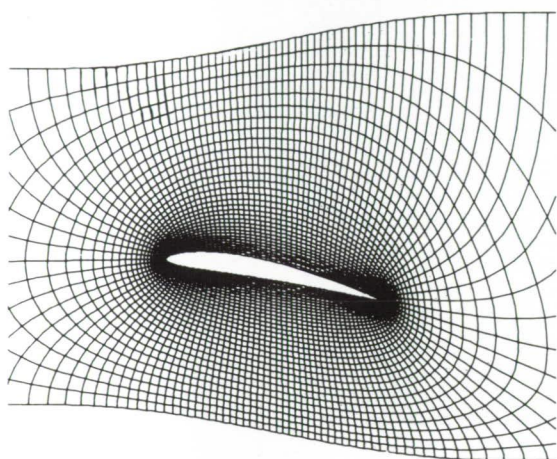
Research on confined or bounded flows enables Langley scientists to interpret experimental results obtained in wind tunnels and may lead to wind tunnel design improvements. This two-dimensional grid was developed to study transonic potential flows in an arbitrarily shaped channel.

The elliptic methods are very capable and can be used to generate grids between boundaries of virtually any arbitrary space. Hyperbolic differential equation methods have been developed which generate grids much faster than elliptic methods. These methods can produce near orthogonal grids, and the control of grid spacing is more direct and effective than it is for elliptic methods. On the other hand, hyperbolic differential equation methods allow only one boundary to be specified so they are only used for grids on which external flows are computed. Methods based on parabolic differential equations are beginning to appear, but they have yet to receive wide acceptance.
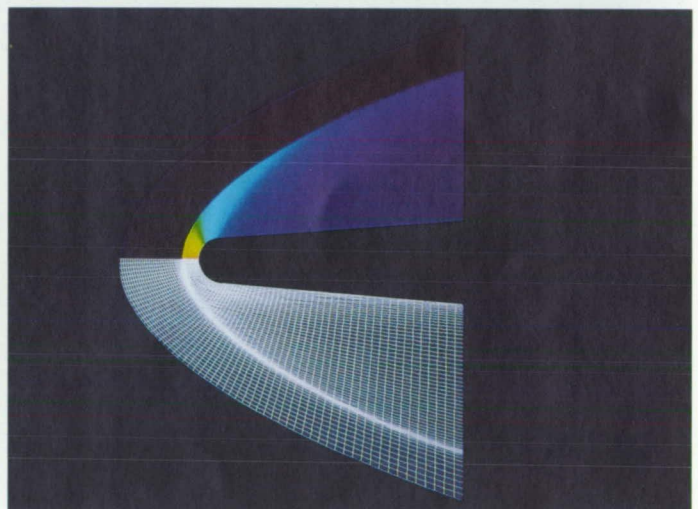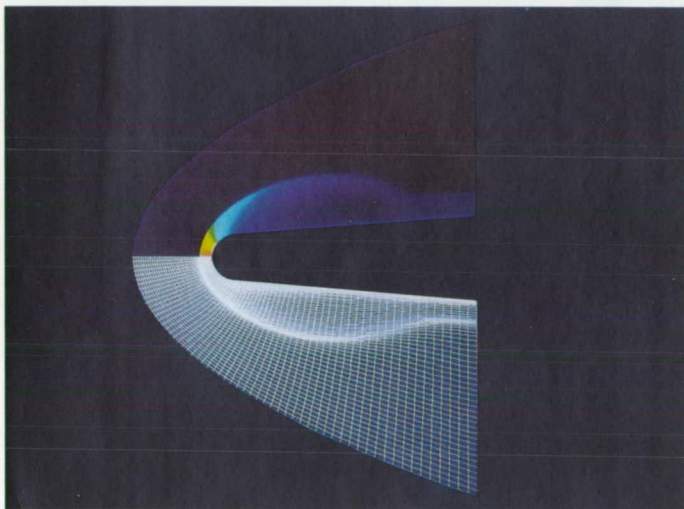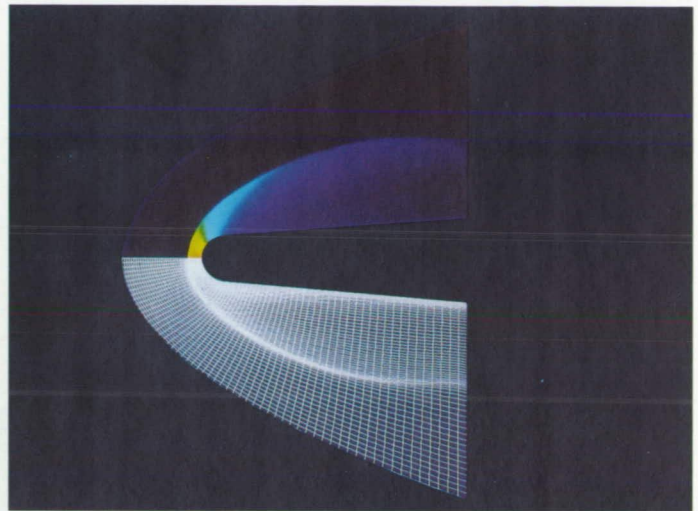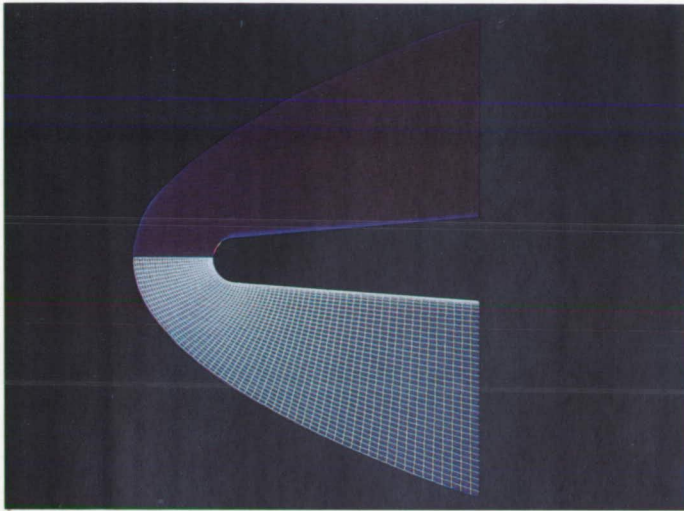
The other approach for generating structured grids is direct algebraic construction. The algebraic methods are basically forms of mathematical interpolation. Two popular methods are the two-boundary scheme developed at Langley and the multi-surface scheme developed under Langley sponsorship. The advantages of algebraic methods are their speed of construction (one can generate hundreds of algebraic grids for the price of a few elliptic grids) and their almost unlimited control of grid point spacing. Because of their speed, algebraic schemes work well in an interactive graphics environment.

Because structured approaches generate actual coordinate systems, and because the computational domains are always rectangular with unit spacing, the CFD applications code itself can be independent of the grid generation method. Most CFD application codes in use today are based on structured grids and can be used with grids generated by either differential or algebraic methods.

To apply structured grids to complex geometries such as complete aircraft configurations, it is usually necessary to divide the flow domain into several blocks or zones, where structured grids are generated within each block. The CFD applications code must then deal with the interfacing of the fluid solution between zones. This is not always an easy task, but significant progress has been made in this area. Most new CFD application codes provide for block grid structure.

For extremely complex flow domains, structured grids become untenable, requiring large numbers of blocks. To surmount this difficulty, CFD researchers have begun to develop unstructured grids and corresponding CFD application codes. Unstructured grids have been used for many years in finite-element analysis of physical structures. However, most structural analysis uses linear partial differential equations; and the solution techniques are well-defined and, relative to CFD, require little computational time. Since almost all of CFD deals with nonlinear partial differential equations, the application of the finite-element method and unstructured grids has not received much attention until recently.

Unstructured grids are usually composed of triangular area elements in two dimensions or tetrahedrons in three dimensions. These grids are naturally body-conforming, but the grid points do not have a consistent underlying relationship. A relationship must be explicitly created through a table that denotes how the grid points are connected. The process of consulting the table every time a grid point is used and the additional storage required for the connectivity table are disadvantages of unstructured methods. On the other hand, the creating of additional grid points (triangles or tetrahedrons) does not require recomputing the entire grid — only the location and updating the connectivity of the additional points are required — and this can be very important for adaptive grids.

Research on unstructured grids and associated CFD application codes is progressing at Langley and at other locations under Langley spon-

By adding additional grid points or, as can be seen here, by changing the distribution or grid points, researchers can focus on a particular area in the flow during the course of the calculation and increase the accuracy of a solution.

ORIGINAL PAGE
COLOR PHOTOGRAPH

Unstructured grids can conform to such complicated structures as complete aircraft. By using an unstructured tetrahedral grid, researchers under Langley sponsorship determined the pressure generated on a complete Boeing 747 flying at M = 0.84. Because unstructured grids require more computer resources than structured grids, current research is focused on overcoming this restriction.

sorship. Unstructured grids are beginning to play an important role in computing flows about very complex geometries. An example is the Euler flow solution about a complete Boeing 747 aircraft under transonic flow conditions.

When a grid is generated and does not change throughout the course of a CFD calculation, the grid is said to be fixed. If the grid points are allowed to move in response to the solution, or if additional grid points are created during the progress of a solution, the grid is said to be adaptive. It is the ultimate goal of CFD researchers that all grid generation be adaptive.

An important factor for an adaptive grid is the criteria for increasing or decreasing the space between grid points. It is usually in the form of a weight-function measured over the entire grid and determined from current CFD calculations. The weight function is small where the grid spacing should be large, and it is large where the grid spacing should be small. The weight function is a linear combination of user specified parameters and computed variables such as numerical truncation error, pressure gradient, etc.

For structured grids the adaptation is usually in the form of grid movement. The number of grid points is fixed and based on the weight function, and a new distribution of grid points is computed. Successful adaptive grid schemes have been constructed with both differential and algebraic techniques in two dimensions. The challenge today is to extend these ideas to multiple-block grids in three dimensions.

A second approach to adaptive grid generation is merely adding additional points to regions where the weight function indicates more points are necessary. This works best in the structured environment when entire blocks of finely-spaced points are added; and in the unstructured environment, additional points only change the connectivity table.
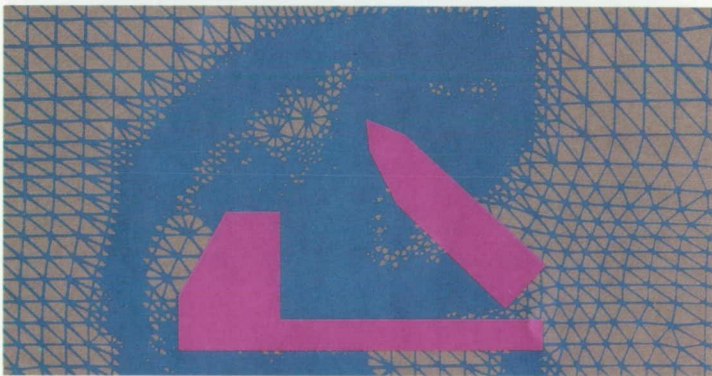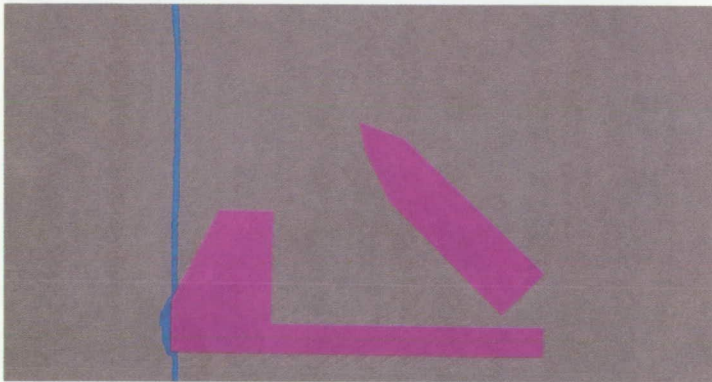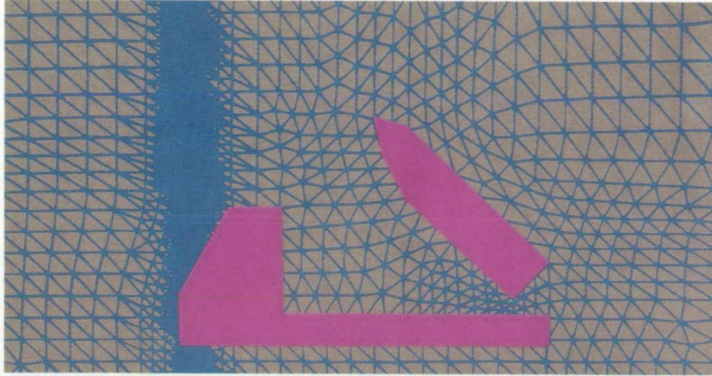
An adaptive grid automatically adapts itself to provide the most points and details where gradients are strongest. These images show an adaptive grid changing as a M=10.0 shock wave travels over an obstacle (work supported jointly by Langley and Naval Research Labs).
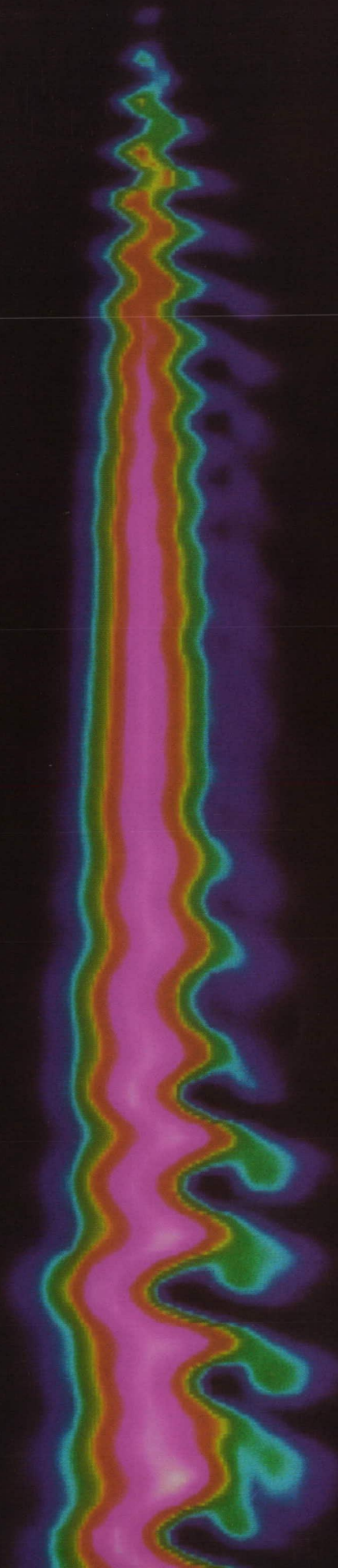
TRANSITION & TURBULENCE

Most fluid flows begin in a laminar (or smooth) state, and then, due to a variety of physical instabilities, undergo a transition to a turbulent (or chaotic) state. Nearly all flows of technological importance contain significant regions of transitional and turbulent flow. The dynamics of these regions are influenced by extremely small spatial and temporal scales that cannot be computed directly using the Navier-Stokes equations with any existing or foreseeable supercomputer. Current practice is to treat the turbulent regions in an approximate sense, using the so-called "Reynolds-averaged" Navier-Stokes equations. The averaging process used in deriving these equations introduces additional terms into the equations which are not known in closed form and must be modeled. The current state of turbulence modeling is still primitive, and this poses a major barrier to improved CFD solutions.

A major research effort has been underway for some time at Langley and elsewhere to comprehend the basic physical processes occurring in transitional and turbulent flows. These efforts are focused on flows in simple geometries and under less strenuous conditions than are present in true aerodynamical situations.

In the past several decades, intensive experimental investigations of basic flows have yielded highly plausible qualitative pictures of the main events in transitional and turbulent flows. Experimentally, however, relatively few flow field properties are accessible. Numerical simulations based on the (non-averaged) Navier-Stokes equations have recently become feasible for these basic flows. Such simulations provide the researcher with the complete flow field. This enables microscopic investigations to be performed for the study of physical mechanisms and also enables the researcher to gather precise
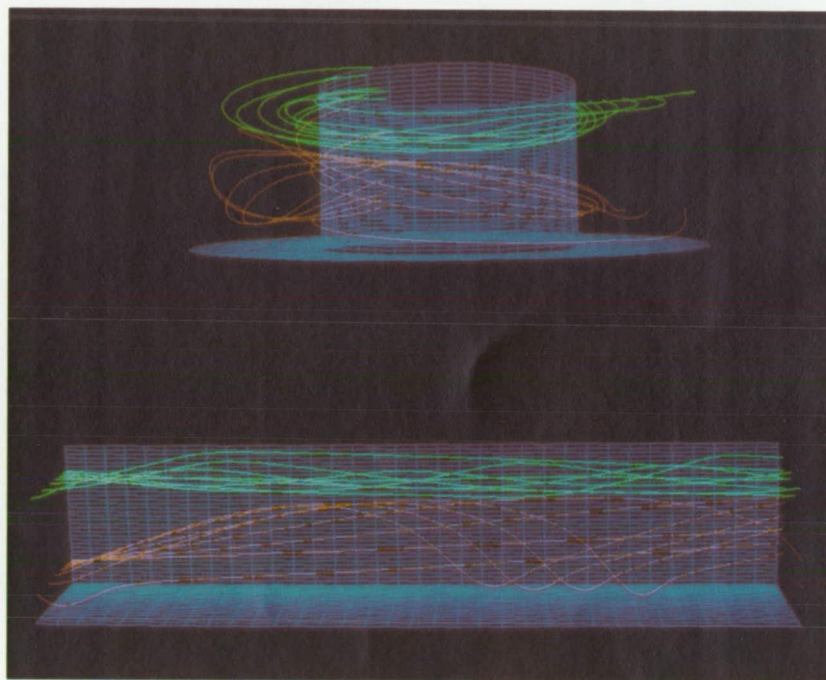
By studying turbulent reacting flow fields, Langley researchers can gain a better understanding of combustion processes and improve aircraft propulsion system efficiency. This image (left) shows the resulting product (water) of a reacting supersonic flow of air and hydrogen gas; the resulting vortical rollup of fuel and air (called unmixedness) retards combustion efficiency.

Taylor-Couette flow is the flow produced when one cylinder rotates with respect to another. Here, Langley researchers have calculated streamlines that originate near the top and bottom regions of the cylinders and have shown them in three dimensions (above). The top image shows the flow around the inner rotating cylinder (the stationary outer cylinder has been removed). The bottom image shows the same flow after the inner cylinder has been unwrapped to appear as a flat plane.
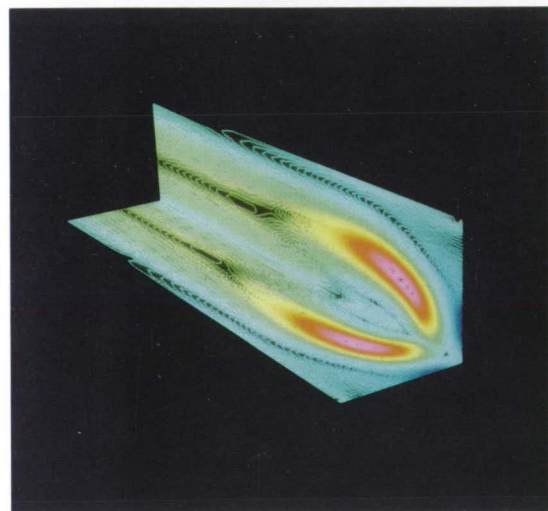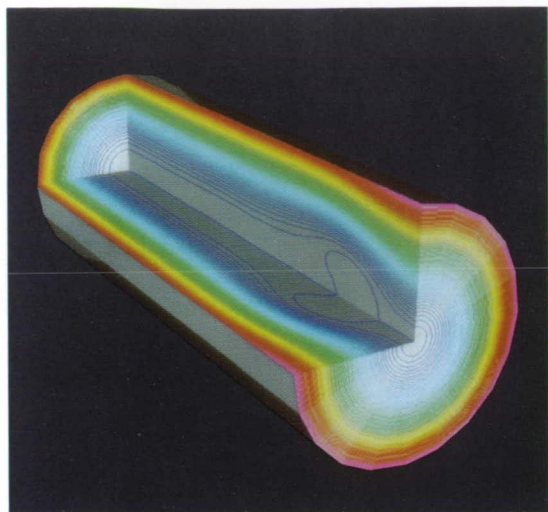
Langley researchers compared two different methods for modeling incompressible steady axisymmetric flows. Euler equations proved inadequate to represent the physical phenomena of vortex breakdown. However, results from the Navier-Stokes equations (shown) agreed with the experimental trends and previous computational studies. These images show the streamlines (top) and vorticity (bottom) for flow through a long cylindrical tube.

information on any term in the equations. Thus, the basic physics can be explored, and modeling efforts can be tested precisely. The understanding and models developed by studying these simple flows can hopefully be extended to the complex environments.

Langley's current research program in transition and turbulence has several components involving direct simulation. They include: improving and extending numerical simulation techniques; application of numerical simulation to incompressible, compressible, and reacting flows; and interpreting the results of numerical simulation. Researchers are conducting this basic research in support of several major areas of aeronautics at the Center. A few of these are laminar flow control technology, supersonic and hypersonic transition, turbulence management and control, and supersonic combustion for hypersonic propulsion.

An early goal of numerical simulation of transition and turbulence was to verify that the correct physical phenomena were indeed reproduced. Now that researchers have established qualitative correspondence between simulation and experimental observations, they are directing their attention toward quantitative comparisons and are focusing on the basic mechanisms. Langley researchers have reproduced the experimentally observed flow structures in channels and flat-plate boundary layers in numerical simulations. They are now engaged in detailed quantitative studies of several types of nonlinear instability mechanisms to establish amplitude thresholds across the speed range, the relation of these mechanisms to later stages of the transition process, and their sensitivity to such laminar flow control techniques as heating or cooling, pressure-gradient and suction. Langley researchers have already discovered a new type of nonlinear instability.

Very little is currently known about the physics of transition in high-speed flows. Langley scientists are proceeding warily from the known grounds of low-speed transition. They are studying the effects of compressibility on the instability mechanisms that have been identified for incompressible transitional flows. They are also beginning to investigate purely compressible mechanisms.

Turbulent, as opposed to laminar, flow can be either bad or good, depending upon the circumstances. In general, it is preferable to have laminar flow on the surface of aerospace vehicles – it has lower surface drag and surface heating. In wall-bounded shear flows, turbulence persists in the flow field downstream of the transitional zone. The maintenance of the turbulence is thought to be a function of the rough balance between turbulence production near the wall (largely in the form of turbulent bursts) and turbulence dissipation due to viscosity. The principal goal of Langley's viscous drag reduction program is to devise means of altering this balance by locally suppressing turbulence production using such devices as riblets and large-eddy breakup devices (LEBU's).
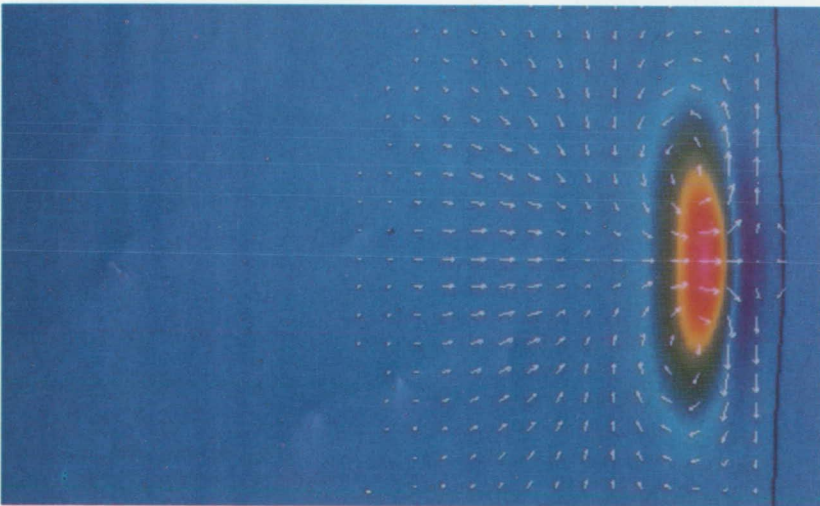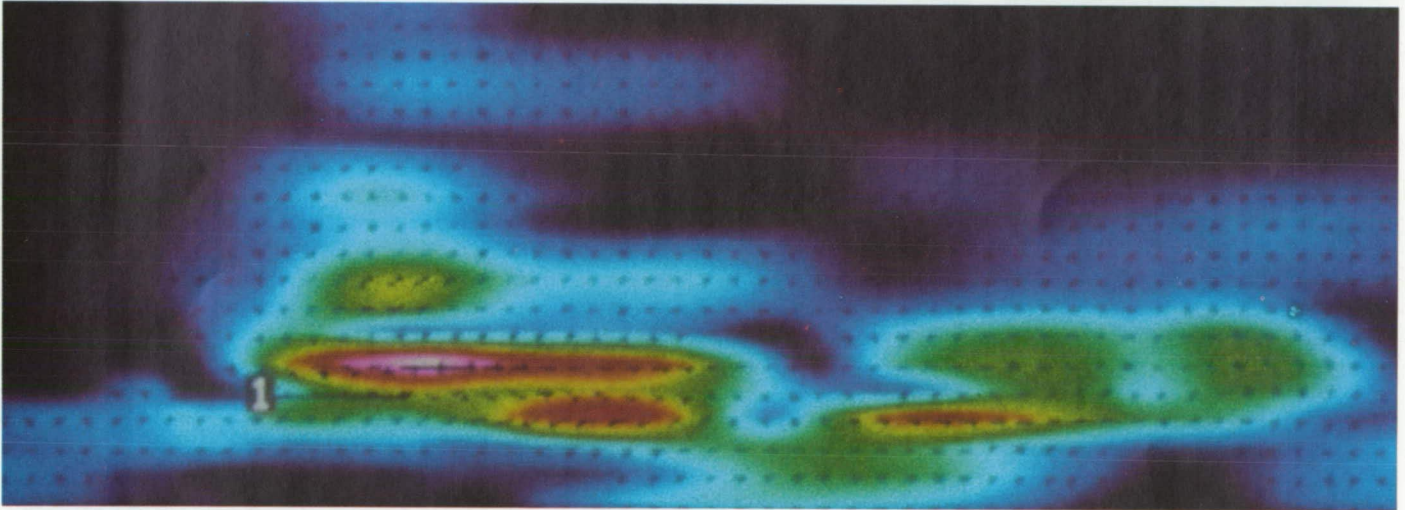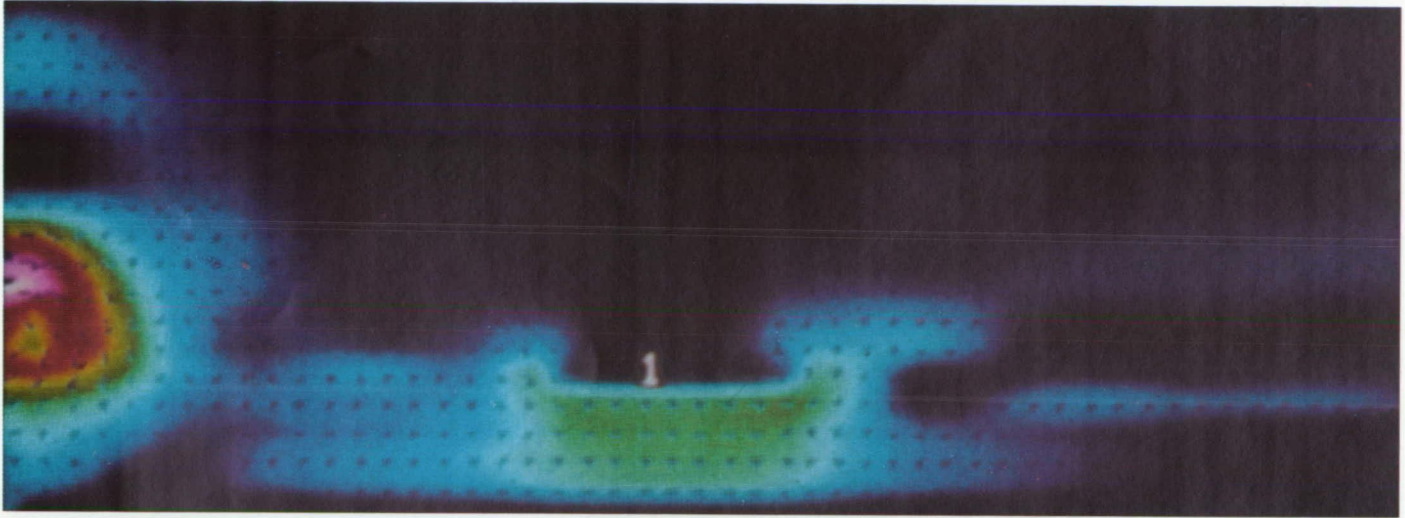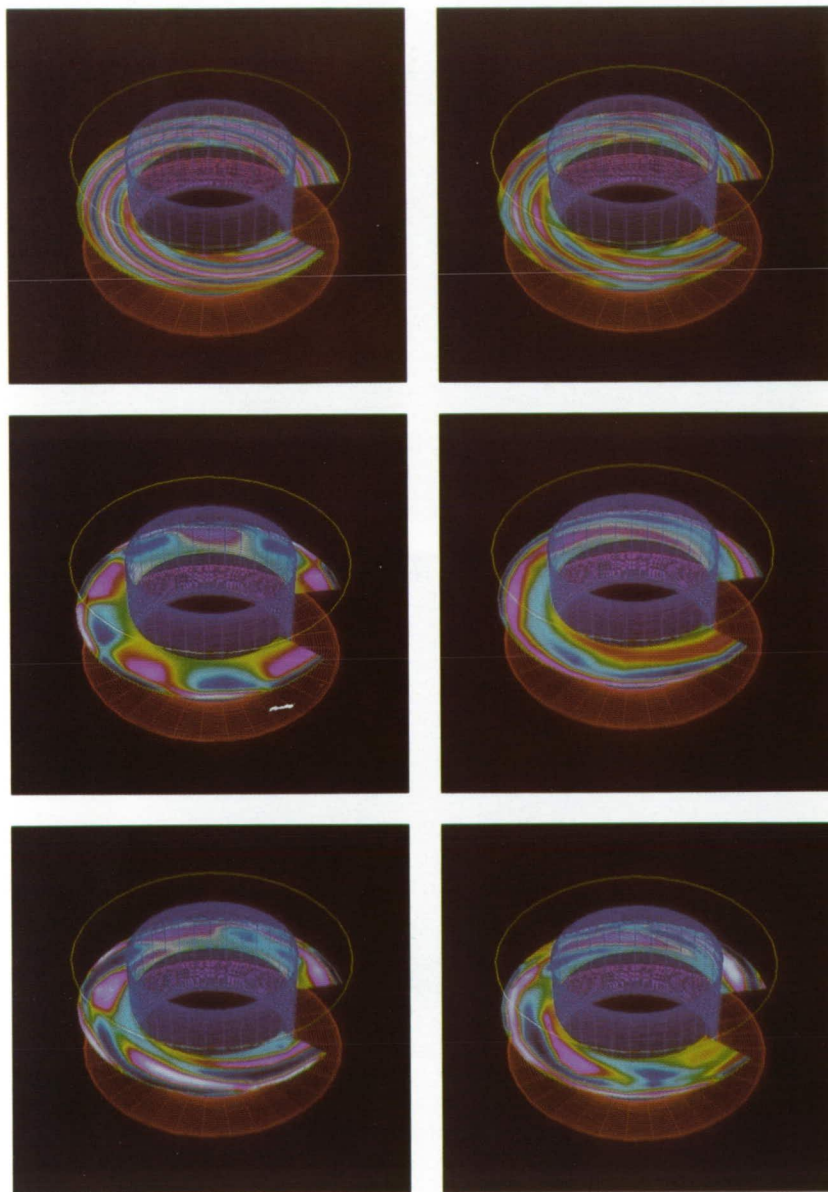
In supersonic flows, turbulence is generated when shock waves encounter hot and cold spots or existing turbulent structures. The critical effects occur at the narrow shock front and play a crucial role in the dramatic increase of turbulence behind shock waves. This image (left) shows a shock wave (black line) moving from left to right across an initially circular hot spot in the flow field; the velocity vectors and entropy contours show two vortices forming as the hot spot becomes highly compressed.

Developed by researchers under contract to Langley Research Center, the Spectral Element approach overcomes limitations in modeling turbulent flows. It promises to be an effective tool for studying Large Eddy Breakup devices (LEBUs), enabling researchers to reduce drag and yield aggregate savings of $200 milion per year in U.S. aircraft fuel consumption. These figures show a vortex (white, red, yellow) moving past an LEBU.
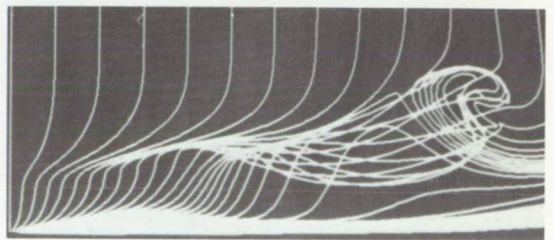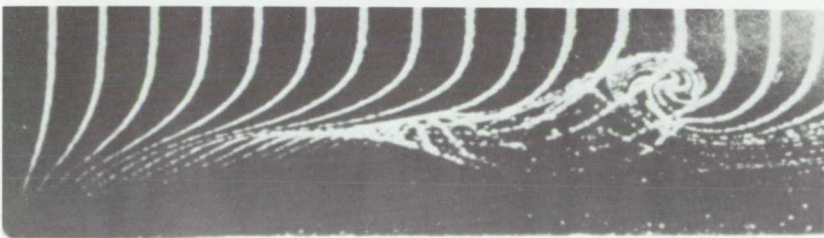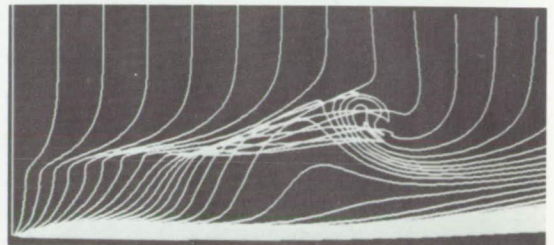
33

In some instances, high-speed turbulent boundary layers are amenable to low-speed turbulence modeling approaches. However, these fail near strong shock waves and in regions of separated flow, even when obvious effects of density changes are incorporated. Researchers have formulated a model of the shock wave effects in the mathematical context of the compressible Euler equations. These are solved numerically with consistent checks on the resolution of the pertinent scales. This model problem is now used to provide a test bed for advanced models of turbulence.

In contrast, turbulent flow is highly desirable in combustion problems, since chaotic fluid motions enhance the mixing of fuel and oxidizer. Since the extraordinarily harsh environment in the flame makes accurate measurement of flow data difficult, the details of supersonic reacting flows are difficult to understand. Langley researchers are conducting a joint theoretical/experimental effort to understand the fundamentals of supersonic combustion. Non-intrusive laser diagnostics are being developed to study non-premixed supersonic flames in simple geometries; comparison with numerical experiments are also underway. Initial work is focusing on non-premixed supersonic shear layer flames. The goal is to develop a picture of the flame instability mechanisms, as well as the flame structure. Researchers are also conducting studies of the interaction of shock waves with flames, along the lines of the shock-turbulence interaction studies.
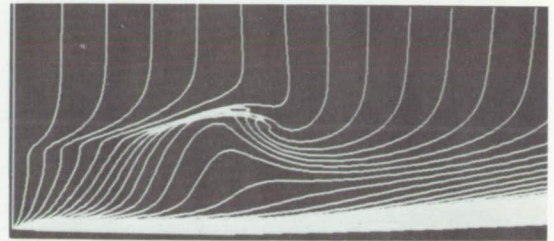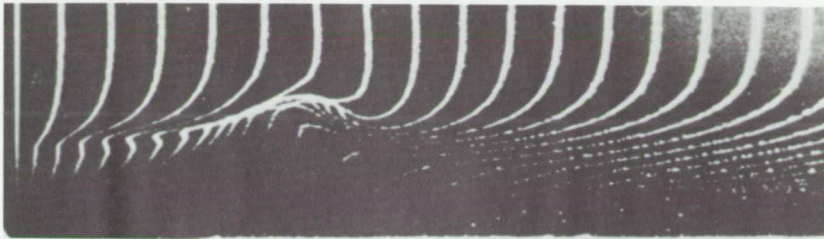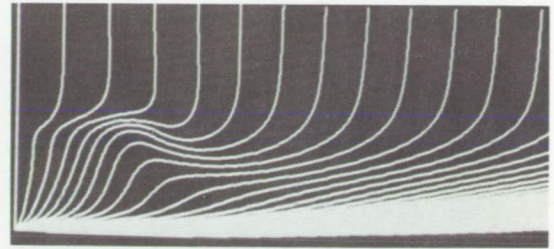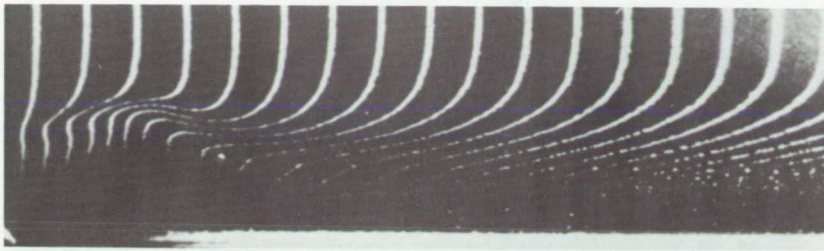
Mode bifurcation and chaos are believed to be important processes in some transitional flows. Taylor-Couette flows are rich in such phenomena; numerical simulations of this model flow are being used to investigate the underlying physical processes. Here, color contours of the axial velocity component taken on a plane midway between the apparatus endplates show the sequence of oscillatory modes taken by the flow with increasing Reynolds number; the bottom figures show a fully chaotic flow.

These images demonstrate Langley's advances in the numerical computation of transitional flows. Photographs of an actual experiment (left) show the paths of hydrogen bubbles released into an airstream, defining the transition of the air flow from laminar to turbulent. The numerical simulations (right) obtained by Langley researchers show strikingly similar results.

RAREFIED CFD

There has been a substantial increase in the need for CFD studies of rarefied gas flow in the 1980s. Until recently, there has been no effective method available to study these flow field problems. In a typical rarefied problem, a number of gas sources contribute to the flow field. They include outgassing from vehicle surfaces, rocket exhaust plumes, waste gas venting and leakage, and the ambient atmosphere. Molecules from each source interact with themselves and molecules from other sources, as well as the vehicle solid surfaces, to form a complex flow field. Generally, molecular collisions affect a large portion of the flow field. These collisions can produce changes in internal and translational energy, molecular composition and radiation. At high altitudes, there may be so few molecules that the whole concept of continuum becomes invalid. The Navier-Stokes equations do not apply in this case, so researchers employ a technique called "Direct Simulation Monte Carlo" (DSMC) particle dynamics.

Combined with major advances in computational capabilities, this technique has led to a practical, economic method for studying flow fields in the vicinity of complex geometry spacecraft. The DSMC method can consider non-equilibrium effects (a measurable "lag time" for the occurrence of chemical and molecular reactions), chemistry and radiation. Here, the physical properties throughout the flow field are computed by calculating molecular entries into the flow field from all sources (both external and internal boundaries), molecular trajectories, molecular collisions, surface collisions and molecular exits from the flow field. The flow field is divided into a large number of computational cells that have dimensions less than the local mean free path (the average distance traveled by a molecule before colliding with a neighbor). A large number of modeled molecules, each representing a very large number of physical molecules, is used to analyze the real gas flow. Scaling down the freestream density by a large factor has the effect of substantially reducing the number of molecular trajectories and molecular collisions that must be calculated. The DSMC method is being used successfully to study a variety of conditions and phenomena experienced by the Space Shuttle and other aerospace vehicles.
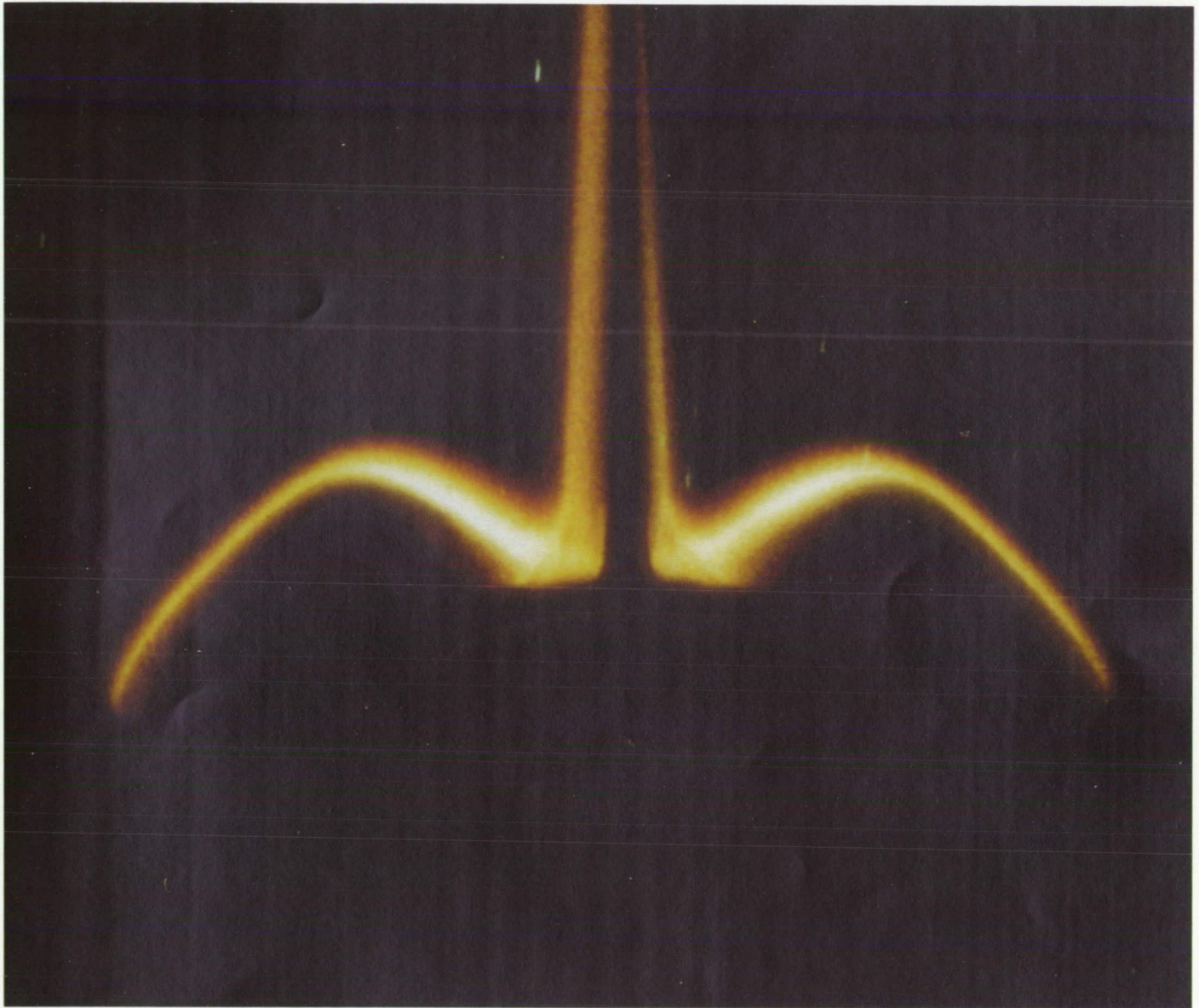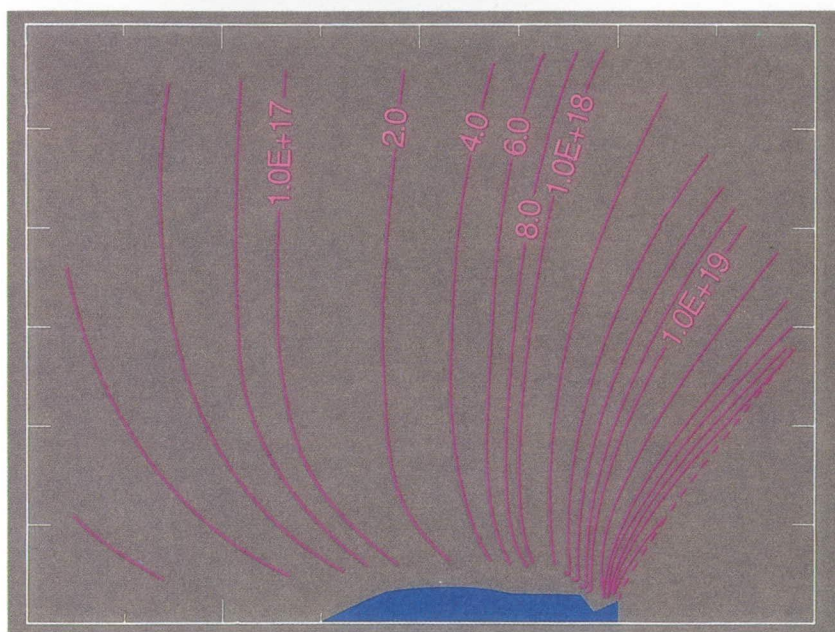
S. B. Mende, Lockheed Palo Alto Research Laboratory
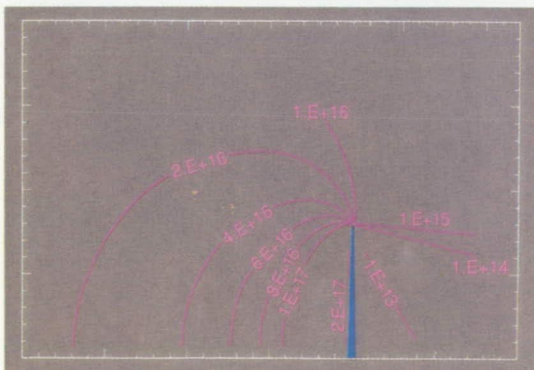
Shuttle crews observed an optical glow around the Shuttle's vertical fin (above) during flights. Researchers are using the Direct Simulation Monte Carlo method to study this mysterious phenomenon, analyzing the gas density, velocity, flow angle and collision frequency of the rarefied flow field to determine the mechanisms that produce this effect. This image (left) shows the calculated density of the flow field around the Shuttle fin.
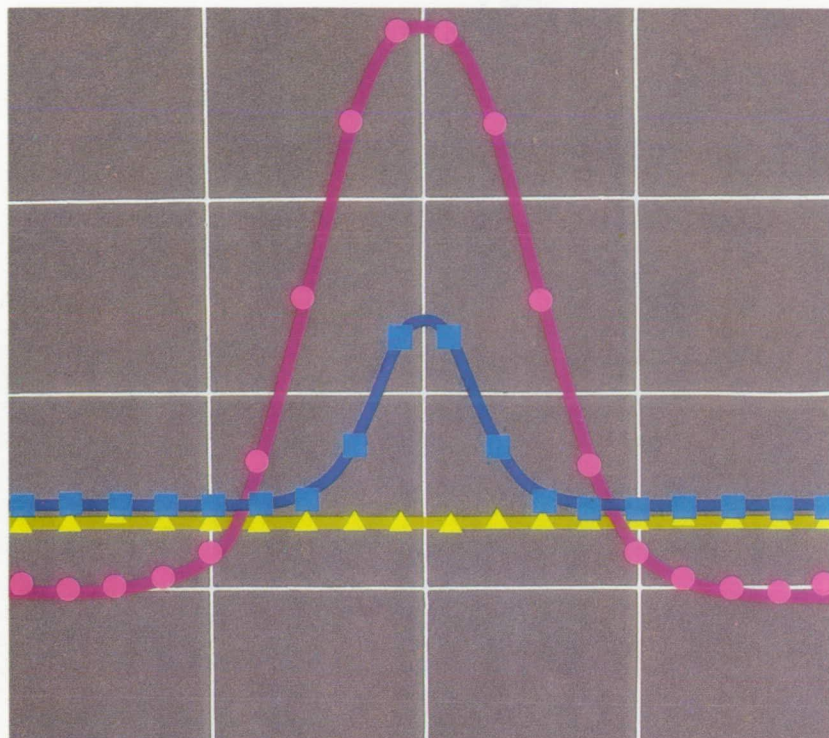
In entry problems, equilibrium assumptions (i.e., infinitely fast reactions) are not valid over a large portion of the flight path. Even though peak heating and primary maneuvering for most hypersonic vehicles will occur under continuum conditions, some portion of the vehicle may experience rarefied transitional flow, particularly the leading edges or nose of the vehicle. The term "transitional" here means the flow or density condition between the "continuum" flow, where the Navier-Stokes equations apply, and the very rarefied "free molecule" flow, where molecules seldom interact with one another. For transitional hypersonic flow, the numerical simulation is complicated, because there is usually an absence of equilibrium among the various energy modes (translational, internal and chemical). Currently, there appears to be no alternative to the Direct Simulation Monte Carlo Method. Researchers are using the DSMC method to obtain further insight into the nature of the flow and the aerodynamic characteristics of blunt, slender bodies under hypersonic transitional flow conditions.

Researchers are also using the DSMC method to calculate both the external flow around the nose region on the Shuttle Orbiter and the internal flow in an inlet tube that provides the initial path between shock-processed gases and a mass spectrometer mounted inside the Orbiter. The objective of the Shuttle Upper Atmospheric Mass Spectrometer (SUMS) experiment is to determine the freestream density based on the mass spectrometer measurements in the altitude range of 160 to 190 km (essentially the transitional flow region). Numerical simulation studies are used to relate inlet pressure to freestream conditions. Along with the procedures for relating the external and internal flows, a three-dimensional version of the DSMC method has been used to simulate the inlet tube flow.

This same method is now used to study phenomena in the vicinity of the Shuttle vertical fin. Crew observation of an optically contaminating glow near Shuttle surfaces exposed to the atmospheric stream (especially on the vertical fin and OMS pod) have led to theoretical and experimental studies to identify the physical mechanism involved and determine the source of this radiation. The DSMC approach is an important tool among computational techniques available to aerodynamicists, particularly as the operating en-

Designers must contend with rarefied flows in space that cannot be calculated with conventional computational methods. The firing of the Inertial Upper Stage (I.U.S.) unit that boosts Shuttle cargoes into high earth orbit (top) produces rarefied gas flows that can contaminate and affect sensitive subsystems. The Direct Simulation Monte Carlo (DSMC) method is enabling Langley researchers to study the flow of these gases. This figure (above) shows the flow about an I.U.S. during firing; it was calculated with the DSMC method.

Numerical analysis of the flow helps scientists to understand what happens to flows directed to experimental instruments and to verify results obtained from Shuttle experiments. This graph shows side wall pressures on an instrument inlet tube; it was obtained through a three-dimensional Direct Simulation Monte Carlo method.
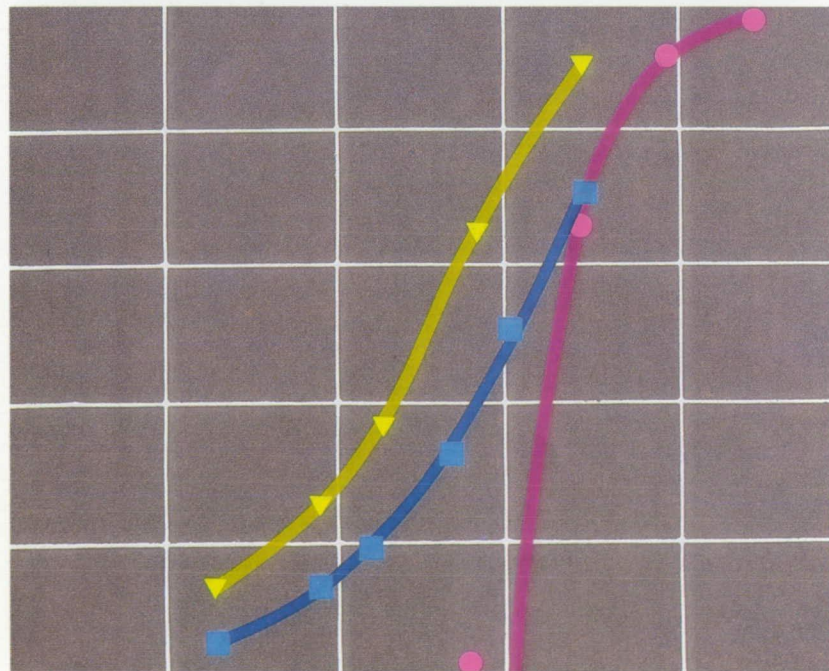
There are significant differences between DSMC methods and conventional computational fluid methods at high altitudes and speeds. The graph's circular symbols show heating coefficients obtained through conventional methods for a vehicle traveling at 18,000 mph at altitudes ranging from 30 to 60 miles. The square and triangular symbols (for two different geometries) at the same speed and altitude were derived with DSMC methods; they more closely follow the physics that occur at high speeds and altitudes.

velope of American transport craft moves higher and higher into the space environment.

Langley Research Center is committed to continuing its research programs in computational fluid dynamics. Advances in computer hardware and software, in mathematical equations and numerical simulation, in grid generation, and in effective studies on fluid transition and turbulence are all contributing to the development of scientific, industrial and governmental aircraft and spacecraft. With advances in computational fluid dynamics, the work accomplished at Langley is enabling scientists and engineers to design fast, safe, highly maneuverable aircraft and spacecraft, pioneering economical testing procedures and speeding the design process.