ICAS-88-4.8.1

NAS - THE FIRST YEAR

F. R. Bailey* and Paul Kutler†
NASA Ames Research Center
Moffett Field, CA 94035

Abstract

This paper discusses the capabilities of NASA's Numerical Aerodynamic Simulation (NAS) Program and its application as an advanced supercomputing system for computational fluid dynamics (CFD) research. First, the paper describes the NAS computational system, called the NAS Processing System Network, and the advanced computational capabilities it offers as a consequence of carrying out the NAS pathfinder objective. Second, it presents examples of pioneering CFD research accomplished during NAS's first operational year. Examples are included which illustrate CFD applications for predicting fluid phenomena, complementing and supplementing experimentation, and aiding in design. Finally, we discuss pacing elements and future directions for CFD and NAS.

Introduction

The role of computers and computational fluid dynamics (CFD) as aeronautical research and development tools is growing steadily in importance. A major pacing item in the advancement of CFD has been the availability of increased computer speed and memory. This fact was recognized over a decade ago by Chapman, Mark, and Pirtle in their article on the comparison of computers versus wind tunnels for aerodynamic flow simulations.(1) Their forward-thinking views gave rise to investigations at NASA's Ames Research Center into the development of advanced computer technologies in order to hasten the advancement of CFD. These initial efforts resulted in the Numerical Aerodynamic Simulation (NAS) Program which, after 9 years of technical study and advocacy, became a NASA new-start program in 1984.(2)

In March, 1987 the program achieved it first major milestone, the Initial Operating Configuration. Today, after a year of operation, NAS provides a large scale simulation capability that is recognized as a key element of NASA's aeronautics program, augmenting both theory and experimentation. NAS provides a world-class supercomputing capability which is readily accessed by the nation's top aeronautical researchers in government, industry, and academia, and offers them the opportunity to perform high-speed computations and simulations for a broad range of aerospace research applications. In addition, the Program serves as a pathfinder in the application of newly emerging computer technologies to CFD and other computationally intensive scientific disciplines.

Computational fluid dynamics is the major application driver for the NAS with the goal of achieving detailed flow simulations about realistic aerospace vehicle configurations throughout the flight envelope. The current NAS system has been used on many pioneering CFD applications during its first year of full operation. Use of the system has grown to over 700 researchers with over 270 projects at 75 locations across the United States. This paper describes the NAS current computational capabilities, its pathfinding achievements, examples of pioneering CFD results achieved on the system during its first year, and further directions for CFD and NAS.

NAS Processing System Network

The first major objective of the NAS Program is to provide a leading-edge computational capability for the United States aerospace community. This is a continuing objective that was first reached in March 1987 with the dedication of the Initial Operating Configuration. The NAS computer system, called the NAS Processing System Network (NPSN), is a large network of computers shown in Figure 1. The key feature of the system is the Cray-2 supercomputers, designated as High Speed Processor number one (HSP-1). They represent the first in a series of increasingly powerful supercomputers.

The production system includes two Cray-2 computers and an ETA-10, model Q; an Amdahl 5880 mainframe computer providing both support processing and access to 300 gigabytes of on-line mass storage; four Digital Equipment Corporation VAX minicomputers; Convex C-1 and Alliant FX-8 mini-supercomputers, and 42 Silicon Graphics IRIS workstations like the one shown in Figure 2. There is also a Thinking Machines Corporation 16,000 processor Connection Machine (CM-2) that is used as a testbed for research in massively parallel algorithms

All the computer systems are linked via Ethernet and Network Systems Corporation HYPERchannel local area networks and are also linked to other computers both at Ames Research Center and at remote locations across the United States. The entire NPSN operates under the UNIX^{tm} operating system with DOD Internet (TCP/IP) network communications provided via the well known Berkeley UNIX "r" commands. Thus, a user can access any computer, run jobs, and transfer data among computers using a single set of commands on all computers. A more complete description of the NPSN is given in reference (3).

Remote users access the NPSN through a wide choice of existing national communication networks including the Department of Defense sponsored MILnet and ARPAnet, the National Science Foundation sponsored NSFnet. NAS is also a member of the Bay Area Regional Research Network that provides 1544 kbits/sec service among Ames Research Center, Stanford University, and the University of California campuses at Davis, Berkeley, San Francisco, and Santa Cruz. MILnet is the main communication path to Department of Defense laboratories, while ARPAnet, NSFnet, and BARRnet serve the university community. For NASA and aerospace industry users, NAS has developed NASnet which is a unique, NAS specific network using Ethernet bridging technology to provide fast response and high throughput to remote graphics workstation users. NASnet currently serves 31 remote sites at bandwidths ranging from 56 to 1544 kbits/sec.

NAS Advanced Features

A second objective of the NAS Program is to act as a pathfinder in the development of large-scale computer

---

* Chief, NAS Systems Division

† Chief, Fluid Dynamics Division

capability through integration of state-of-the-art improvements in hardware and software technologies. In carrying out this objective, the Program has been guided by three principal goals. The first is to furnish the most powerful commercial supercomputers as soon as they become available and to provide a system that can accommodate change. The second is to provide a common user view and uniform software interface to the entire supercomputing system even as it evolves. The third is to improve user productivity and access.

Supercomputing

One of the NAS Program's first accomplishments was the installation in 1985 of the Cray-2 computer, the first "gigabyte class" supercomputer. The Cray-2 with its memory of 2.048 gigabytes (over 268 million 64-bit words) represents a two-orders-of-magnitude increase over previously available capacity. The Cray-2's very large memory, coupled with its ability to sustain a computing rate of 250 million float-point operations per second, allows CFD researchers to increase their application program size and complexity without significant code modification. The very large memory virtually eliminates memory-disk transfer overhead during execution of large programs. The net effect is decreased preparation time for researchers tackling increasingly complex problems, plus improved job turnaround time. More importantly, researchers are now able to perform fluid-dynamics simulations that were not possible just a few years ago.

Uniform Software Environment

Another pathfinding achievement is the uniform software environment pioneered by the NAS System. NAS was the first supercomputing facility to install a standard operating system and communication software on all processors. Ease of use is enhanced since all NAS computers, including the Cray-2s, operate under the UNIX operating system. UNIX offers users the flexibility of both interactive and batch supercomputing. Moreover, the software system presents a common system interface to the users and provides the same environment, i.e., provides common utilities and commands, on all user-visible subsystems. The uniform environment enables NAS users to move easily between processors, allows for easy file access and command initiation across machine boundaries, and enhances user-code portability within the NAS processing system. Furthermore, the open architecture concept (implementation based on openly available definitions for hardware and software interfaces) used in the design of the NAS system, allows for more modularity and for easier implementation of new capabilities.

Remote Access

NASnet is an example of another pathfinding accomplishment. The importance of ready access by the national aerospace research community has led to the development of NASnet as a unique high performance nationwide communication network that links remote user locations to the NAS facility via land-links, with communications speeds up to 1544 Kbits/sec. At the forefront of Ethernet bridging technology, NASnet provides an environment in which researchers at remote locations (Fig. 3) have virtually the same interactive capability as users at the NAS Facility. One example involves graphical co-processing using the Ames-developed Realtime Interactive Particle-Trace (RIP) application.(4,5) With this application, computational and memory intensive raw result data, such as grid-defined flow-field quantities on the Cray-2, and display and manipulation data, such as fluid particle traces on the remote workstation, are passed back and forth via network links. The combined use of workstation and supercomputer offers the remote user a 10:1 time savings improvement over the workstation alone in performing complex displays of three-dimensional fluid dynamics solutions. In fact, the NAS system is designed so that all users can consider the High Speed Processor as an extension of their own workstation. Furthermore, new users familiar with the UNIX-based workstation environment (such as that offered by Sun Microsystems Inc., Silicon Graphics Inc., Digital Equipment Corp., and others) quickly become productive on the NAS supercomputers.

Graphics Workstations

The NAS Program has emphasized the development of graphics workstations as the principal means for researchers to interface to supercomputers.(6) All of the CFD results presented in this paper were displayed on one of these workstations. NAS pioneered the direct networking of workstations and supercomputers that literally transforms the supercomputer into an extension of the workstation. The user resides at his workstation where text and small data files are created and modified, and where complex input data and results are displayed and analyzed. Interactive graphical post-data processing is an essential aspect of the today's sophisticated numerical simulation process.

Unsteady, three-dimensional numerical simulations generate an enormous amount of data and it is virtually impossible for the scientist to digest those data in a reasonable amount of time and to gain any qualitative picture of the physical phenomenon simulated. Graphics workstations have been employed to aid this dilemma. The workstations typically consist of high-resolution, high-throughput color graphics terminals that enable near real-time manipulation of view point and other display parameters to give enhanced three-dimensionality to the results. The software that operates on these workstations (e.g., PLOT3D, RIP, and GAS designed by Ames researchers(7)) permits scientists to display any facet of their data for easy interpretation and evaluation. Evidence of early success in this area is demonstrated by Boeing Corporation's new Computational Fluid Dynamics Laboratory which is patterned after the successful workstation network pioneered by NAS.

NAS as a Continuing Research Tool

The final major objective of the NAS Program is to provide a strong research tool for NASA's Office of Aeronautics and Space Technology (OAST). The OAST Research Centers at Ames (California), Langley (Virginia) and Lewis (Ohio) are well-positioned to take advantage of the NPSN capability. Langley and Lewis users not only access the NAS system in the same manner as do local Ames users, but also utilize their Ethernet connected workstations (12 at Langley, 5 at Lewis) to access computer resources at their own Centers. High-bandwidth terrestrial communications, 1544 kbits/sec to Langley and 224 kbits/sec to Lewis (to be increased to 1544 kbits/sec at year's end) provide the responsiveness and capacity to effectively perform workstation-to-Cray distributed applications like RIP.

To maintain the lead in large-scale computing capability, NAS is installing a new high-speed processor representing each new generation in a series of increasingly more powerful supercomputers as soon as it is available. This strategy (Fig. 4) began in September 1985 with the installation of the first full-scale Cray-2, as HSP-1. A second Cray-2 was added in January, 1988, and it will be replaced by HSP-2 in late 1988. The new HSP-2 is anticipated to be one of the earliest units of the next generation supercomputer and will lead to a four-fold increase in operational computing performance. The third generation, (HSP-3), is targeted to replace the remaining Cray-2 in the 1990 time frame. This ongoing HSP upgrade strategy assures that the newest and most advanced supercomputers will replace the older of the two installed systems. This will maintain a steady state in which there are always two high-speed processors in use, one of which is the most advanced processor available. This strategy is designed to ensure that America's aerospace community has early access to the most advanced supercomputers available.

NAS User Base

The number of NAS users has increased dramatically since the NAS Initial Operational Capability dedication in March of 1987. Current users number over 700 and the number is steadily increasing. Initial NAS users have provided positive feedback on their increased productivity, particularly in reductions in code development and debugging time attributable to the interactive nature of the common UNIX environment and the NAS graphics capability.

The NAS system has attracted users from a widely diverse research and development base. Although government and DoD users make up the bulk of users, industry and university users currently comprise 25%. This share is expected to increase as the communities' experience grows. The NAS capability is also being used by scientists from a wide variety of disciplines. In addition to aeronautical research, applications include benchmarking computational chemistry codes, turbulence research, modeling the structure and properties of new superconducting compounds, simulating the formation and evolution of galaxies, research on Earth and planetary atmospheres, and climate prediction. The major use, however, is for CFD and this use covers a broad range of applications as exemplified in the next section.

Applications

The overreaching goal of the discipline of CFD at NASA is to simulate in a reasonable amount of time the realistic flow about actual aerospace vehicles and their components at true flight conditions. A reasonable amount of time is approximately 15 min on a state-of-the-art computer system. To attain this goal requires a computer system such as the NAS Cray-2 with its large memory and fast clock time as well as accurate and robust CFD software. The first part of this paper described the NAS computer system. In this section we describe current state-of-the art CFD software and demonstration examples accomplished using the NAS-provided capabilities.

There are three major uses of CFD that have resulted in its widespread acceptance as an alternate aerodynamic-prediction tool. They are to: 1) predict fundamental fluid physics, 2) complement and supplement experimental testing, and 3) aid the aerospace vehicle design process. Examples are presented below that demonstrate the use of CFD in these three areas.

Predicting Fluid Phenomena

Vortex Breakdown

With the development of modern high performance aircraft, including those with highly swept leading edges, has come additional flow complexities including the problem of control of the leading edge vortex. For highly swept wings this vortex plays a crucial role in the development and control of lift.

Extensive study of the vortical flow over a double delta wing has been carried out at the NASA Ames Research Center by Fujii et al.(8) on the NAS Cray-2 by taking advantage of its large memory. The numerical simulation solves either the thin layer Navier-Stokes or Euler equations. A typical result is shown in Figure 5. In this figure, off-surface particle-path traces clearly demonstrate the breakdown of the primary vortex. The fine-grid resolution provided by the large memory of the Cray-2 permitted an accurate description of this physical phenomenon. Note that the vortex from the strake region remains orderly and tightly coiled until about 80 to 85% of the chord, where there is an abrupt change of the flow. The particle traces also indicate the existence of streamwise reverse flow along the core, a clear indication of vortex breakdown. This simulation capability will play an important part in the design and analysis of vortical flows for a wide range of aerospace applications.

Turbulent Boundary Layer

In recent years, considerable progress has been made using the NAS system for the numerical simulation of the physics of turbulence using spectral methods to solve the complete, time-dependent Navier-Stokes equations without small-scale modeling. By employing a high resolution workstation, it is possible to view the development and propagation of the numerically predicted turbulence quantities. Such a result, generated by Robinson and Kline(9) using 9.4 million grid points, is shown in Figure 6 and represents the structure of turbulence in a simulated flat-plate boundary layer. The figure was originally developed in color, wherein each separate element of the boundary layer structure was shown in a different color. In that figure, the elongated white surfaces identified the low pressure vortex cores, the red and blue areas identified where significant contributions to the Reynolds shear stress were occurring, red indicated regions of low-speed fluid being ejected outwards, and blue denoted regions of high-speed fluid being swept towards the wall. From the present figure one can obtain a general idea of the computed results. These kinds of presentations help the scientist to more clearly understand the role that each of the physical variables play in the onset of turbulence, and can significantly aid in the development of statistical models to represent such turbulent flows in Reynolds averaged Navier-Stokes codes.

Jet-Induced Lift Loss on Delta-Wing

With the advent of modern V/STOL aircraft, such as the British Harrier, a great deal of interest has been generated in the fluid dynamics of powered-lift. To study the fundamental mechanisms of such flows, Rao and Van Dalsem(10) who solved the three-dimensional, unsteady, Reynolds averaged, Navier-Stokes equations, simulated the effects of two jets emanating from a 60° delta wing in a cross-flow. The results are shown in Figure 7 for a selected flow condition where the jet velocity is 15 times that of the free stream and the height of the wing above the ground is 0.25 times the span. Using a high-resolution workstation, it is possible to visualize the otherwise complex flow field in great detail. The vortex generated by the interaction of the jet with the cross-flow and the ground plane results in a locally decreased pressure, causing a loss of lift on the wing. In effect the delta wing sees a negative angle of attack and is sucked down to the ground.

Complementing and Supplementing Experimentation

Ogive Cylinder at Large Incidence

Modern aircraft, with their characteristically long, slender nose shapes, and operation at high angles-of-attack, develop unusual flow patterns and forces which frequently affect controllability. Because of this, an experimental research effort by Degani and Zilliac(11) was undertaken to study the unsteadiness of the flow around an ogive cylinder (L/D =12) at 40° incidence. This study was prompted by computations using a three-dimensional, time-dependent, thin-layer Navier-Stokes code in which an unsteady behavior of the flow on the lee side was discovered that produced time-varying normal forces. Closer examination of the solution given in Figure 8 showed what appeared to be shear-layer vortices being shed into the leeside vortex flowfield at a frequency of 1200 hz. Subsequent experiments in a low-turbulence wind tunnel confirmed the flowfield unsteadiness. Figure 8 shows a smoke-laser sheet visualization of the leeside flowfield. High-speed movies of the ripples in the free shear-layer show that they are shed at frequencies near that of the computation. Questions about the origin and structure of the unsteady flowfield are presently being investigated in further detail experimentally using hotwire probes and surface-mounted pressure transducers and computationally using several Navier-Stokes codes.

Transonic Wing-Body Configuration

An example where CFD played a major role in many facets of an experimental research program is the transonic wing-body configuration experiment of Olsen in the NASA Ames High Reynolds Number Channel II facility. As part of the experimental research program to provide data to validate the NASA Ames TNS code (a three-dimensional, zonal, Reynolds-averaged Navier-Stokes program) and investigate the flow about a modern fighter-like, wing-body combination, three separate CFD codes were used. First, a panel-method code was used to evaluate the potential effects of wind-tunnel wall interference, predict the onset of supercritical flow, and evaluate subsonic model loads. Second, an axisymmetric transonic full-potential code was used to redesign the nose of the body to eliminate body-tunnel shock interference and to specify the location of pressure taps to resolve the body flow field. Third, a transonic full-potential wing code was employed to insure the presence of supercritical flow on the wing for the Mach numbers to be investigated, to guide the placement of pressure taps to resolve the wing flow field, and to predict the transonic model loads.

The model is shown in Figure 9, and the experimental results and comparisons with numerical simulations for the wing alone are presented in Figure 10. Future experimental work will include testing the complete wing-body configuration; future computational work will include grid-refinement studies to determine solution independence.

Turbulence Modeling

Current computer power limitations preclude solution of the complete Navier-Stokes equations for flow about realistic configurations, and thus it is necessary to solve a reduced equation set. The Reynolds averaged Navier-Stokes equations, however, require a turbulence model for closure. The turbulence model emulates all scales of turbulence generated in the thin viscous region near the surface of the configuration. Thus turbulence modeling is an important element in the numerical simulation of many flow problems. The development of turbulence models requires a strong interaction between computationalists and experimentalists.

An example of a building-block experiment performed by Bachalo and Johnson(12) for extracting turbulence data, and how it was used in conjunction with computational fluid dynamics to guide the development of a zonal turbulence model for transonic shock wave-boundary layer interactions is shown in Figure 11. The test model consisted of a circular body fitted with a circular-arc section. Transonic flow developed over the circular arc section in a manner similar to that on an airfoil, and shock wave interactions of varying strengths were studied by varying the free-stream Mach number.

Computations of the flow field using a Reynolds-averaged Navier-Stokes code revealed deficiencies in the turbulence model. The experiment used a model developed primarily for attached boundary layers, and the shock wave location was predicted incorrectly. As a result, the pressure recovery was seriously overpredicted. The experimental mean and turbulence profile data were used to explain the differences and guide the development of an improved turbulence model. Using new modeling concepts in conjunction with the experimentally obtained turbulence data, a significant improvement in the results was observed.

Aiding the Design Process

National Aero-Space Plane

The United States has as one of its aeronautical goals to pursue the research and development of the technology necessary to permit an aerospace vehicle to routinely cruise into and out of the atmosphere with takeoff and landing from conventional runways. This effort is called the National Aero-Space Plane program, and will require a hypersonic vehicle to attain the speed of Mach 25 to reach Earth orbital velocity. Computational fluid dynamics is playing a key role in this program because of the lack of ground based experimental facilities to simulate the flow about the vehicle along some parts of its trajectory.

Hypersonic flows, although incorporating the same advantages as supersonic flows, i.e, a limited domain of dependence, contain additional complications that are yet to be fully evaluated. These include more complicated boundary-layer transition and turbulence physics, real-gas effects, merged shock and boundary layers, stronger shock waves, and radiation effects.

Computer software is currently under development to simulate the flow about the plane. The governing equations are the three-dimensional Reynolds-averaged Navier-Stokes. In one approach, the unsteady version of the equations are solved; in another, the parabolized equations (PNS) are solved .

A typical solution about a generic hypersonic configuration using the PNS approach developed by Lawrence and Jettmar(13) is shown in Figure 12. The flow conditions consist of Mach 12.4 and angle of attack of zero degrees. The figure was originally developed in color, wherein a gradation of pressure was depicted by a color scale. The red color corresponded to a high pressure region and the blue color corresponded to a low pressure region. The black lines superimposed on the surface of the vehicle represent streamlines. The shock structure at the aft end of the vehicle was shown by the color band in that vicinity. A high pressure region on the concave undersurface exists in the vicinity of the inlet. This externally compressed flow enters the engine modules where it is mixed with fuel and burned. Comparisons of these numerically generated solutions with experimental data show excellent agreement.

Space Shuttle Main Engine

Operating the Space Shuttle Main Engine (SSME) on the test stand at power settings greater than 104% results in flow problems with components of the liquid hydrogen hot-gas manifold. The flow problems encountered within the SSME, and the subsequent computational analysis and redesign, have served as an indication of the practicality of modern CFD technology to solve real-world problems in an effective manner. Early experience with the SSME indicated that there were problems of reliability and thrust performance, and a large computational simulation research program was undertaken at Ames Research Center by scientists from Rocketdyne (the manufacturer of the engine) and Ames.

A computer code was developed that could treat the complex internal geometry of the SSME powerhead. Kwak et al.(14) developed a three-dimensional incompressible Navier-Stokes code (INS3D) for this application that provided an extensive low-speed simulation capability. Working with engineers from Rocketdyne Corp., this code was applied to simulate the flow within the hot-gas manifold, transfer ducts, and the main injector. Results from this code indicated that the flow within the present three-duct design was inefficient. Large separated flow regions were exposed by the calculations within some of the components, and it was found that the center transfer duct transmitted only 9% of the total flow.

Calculations and cold-flow experiments on a proposed two-duct design showed significantly improved flow characteristics. Figure 13 shows the surface-pressure map for the redesigned powerhead arrangement. As a result of this computational demonstration, Rocketdyne has determined that CFD simulation will be utilized to develop and analyze all future SSME designs.

Space Shuttle Fast Separation

The computation performed by Buning et al.(15) for the time-dependent flow over the integrated Space Shuttle launch configuration consisting of the shuttle, external tank, and solid rocket boosters, is a good example of a contemporary CFD simulation problem. In this case, the numerical simulation is being used to analyze flight performance, and the feasibility of shuttle separation and crew escape during launch. A solution was obtained by solving the unsteady three-dimensional Reynolds-averaged, Navier-Stokes equations using a composite grid and an algebraic turbulence model. A comparison of computed and experimental results are shown in Figure 14. Although the connecting support structure between the Orbiter and the external tank was not modeled, the computed results compare well with the experimental values. Future computational work will extend the simulation to include staging, where the Orbiter separates from the solid rocket booster and external tank assembly.

Elements Pacing CFD

There are two primary elements that are pacing the discipline of CFD. They are physical modeling and computer systems.(16)

With the interest in hypersonic flight, new challenges are facing the CFD scientist.(17) These include developing flowfield simulation codes that account for real-gas behavior. Associated with that is the need for turbulence models for these hypersonic flows. In addition, because hypersonic vehicles cruise at high altitudes, it is important to understand boundary-layer transition and be able to correctly account for it in the simulation tools.

A necessary step, of course, in the development process of any new computer code is validation. Therefore, either experiments or flight tests that can provide data in the real-gas flight regime must be performed to validate or calibrate newly developed hypersonic flow analysis computer codes.

For hypersonic simulations, it is necessary to know the correct rate constants used in the finite-rate chemistry equations that are coupled with the fluid flow equations. In the past, these constants were determined experimentally. However, with the advent of large computer systems and advances in chemical reaction modeling tools. the discipline of computational chemistry is providing accurate rate constants for use in CFD simulation codes.

For the other speed regimes, turbulence modeling remains the pacing element in CFD flow simulations. Until computer systems are large enough to perform direct or at least large-eddy simulations on realistic configurations at flight Reynolds numbers, accurate turbulence models will be required. There currently exists no universal turbulence model and, therefore, nearly each peculiar flow requires its own model. As such, this places an enormous burden on the experimentalist to obtain the necessary data to develop the appropriate turbulence model for a particular flow.

The CFD scientists, as well as many other simulation scientists, are limited by what they can do numerically on existing computer systems. They are limited by the storage and speed available on those systems. Other associated computer limitations include network bandwidths for transmission of data, high-speed mass storage devices for archiving the large data bases generated, and post-processing software and hardware for analyzing the massive data bases generated from unsteady, three-dimensional solutions.

Future Directions for CFD and NAS

A considerable number of challenging technical problems exist for which CFD will be beneficial and, sometimes, mandatory.(18) In the future, CFD will be applied to several interesting and challenging problems that will push the state of the art in the discipline. Typical problems include flows about: 1) powered lift aircraft operating in and out of ground effect, 2) hypersonic vehicles including inlet, engine, and exhaust, 3) rotorcraft in hover, transition, and forward flight, and 4) turbomachinery including pumps, compressors and turbines. To meet these future challenges two areas of particular importance must be addressed. These are the simulation of unsteady viscous flows and multidisciplinary simulation.

The simulation of unsteady viscous flows about realistic aircraft configurations is now possible using computational tools. Several computer codes for simulating these flows have been developed by various researchers. Now we can begin to study unsteady flow problems that result when high-performance aircraft fly at high angles of attack, and to model complex phenomenon such as asymmetric vortex shedding, and vortex breakdown or bursting. The goal is to develop these codes to enable, for example, prediction of aircraft performance near their performance boundaries.

Equally challenging is the ability to integrate several physical models into a single computer simulation program. For example, a typical hypersonic application would include the coupling of the finite-rate chemistry equations with the Navier-Stokes equations. In the past, simple couplings have occured such as linking a structural response code to a fluid dynamics code to study aeroelastic problems, or coupling a flow code with an optimization routine for wing design. The future holds promise for the birth of a new and more computationally intensive discipline called Computational Aeroscience. With more complete coupling, this new discipline will allow the simulation of entire aerospace vehicle systems involving the complex integration of fluid dynamics, propulsion, structural mechanics, thermo-gas dynamics, electromagnetics, and controls.

A major pacing item in meeting these challenges is computer system speed and the ability to use this speed. The achievement of practical, unsteady, fully viscous simulations and mutidisciplinary aeroscience simulations will require approximately 1000 times more computing power than is now available. Furthermore, future simulations and computer systems are becoming so complex that new more sophisticated software tools must be made available if these powerful systems are to be fully utilized. Two promising technologies that can be applied to meet these challenges are massively parallel processors and artificial intelligence (AI) systems.

Most of the CFD development to date has taken place on single processor computers. With advances in clock speed and memory size (brought about largely by more efficient chips) the overall processing speed and power has increased by orders of magnitude during the past ten years. The physical limits of such development are now being approached, caused by inherent electron speeds and problems of cooling.(19) Although new opportunities may be provided by the development of practical means of achieving near-room temperature superconductivity, the most promising future direction for satisfying the insatiable thirst of scientists for CPU cycles is parallel processing.

Most of the current supercomputers have more than one processor, up to eight in the case of the recently announced Cray YMP. Conventional supercomputer technology is likely to increase in parallelism at a moderate rate with steady factors of 3 or 4 improvement in performance. The conventional designs are being challenged, however, by massively parallel architectures offering thousands of processors and the potential for orders-of-magnitude improvements in the near term. The impediment to the rapid application of massively parallel computers to CFD problems is the limited amount of system software currently available. How to formulate the equations of CFD into algorithms suitable for solution on a multi-thousand processor machine, as well as how to manage the data internally among the processors, are first-order research problems at the present time. Nevertheless, progress is being made.

At NASA Ames, the Navier-Stokes equations are being solved on Thinking Machine's 16,000 processor Connection Machine computer (It is designed for 64,000 processors) installed in the NAS system. Great strides remain to be made in this area, but the promise of a new generation of supercomputers, able to handle problems that are now not amenable to practical solution, is very exciting.

A fully productive scientific user interface, needed to meet the dual complexities of massively parallel systems and complex simulations, does not exist and must be developed. What is being proposed at Ames is a computational fluid dynamicists workbench containing graphical, numerical, and symbolic computing capabilities that can be integrated into the NAS-like environments. The techniques and potential capabilities of artificial intelligence (AI) offer a promising direction for the development of the workbench and CFD applications. Expert systems are already being explored to help the CFD scientist divide a complex flow region into several zones, each of which can then be discretized using relatively simple grids. At the other end of the numerical simulation process, visualizing the computed results, AI will be used to investigate and analyze the resulting flow patterns, and identify and display those deemed to be of most interest. Such tools will provide the aerodynamicist with the means to quickly identify problem areas, speed the solution process, and possibly even suggest alternative configurations. Expert system programs will also be developed to aid non-CFD scientists learn to use complicated flow-simulation programs.

Summary

The NAS Program has completed its inaugural year as a premier supercomputing capability serving the aeronautics research community throughout the United States. Several examples have illustrated the power of NAS and CFD to advance the state-of-the-art in predicting fundamental fluid physics, complementing and supplementing testing, and aiding the aerospace vehicle-design process. As part of its pathfinder mission to advance computational capabilities, the NAS Program is now moving ahead to pioneer the introduction of the next generation supercomputer to the research community. However, the Program must meet even greater challenges brought about by the demands of important applications such as unsteady viscous flow simulations and the multidisciplinary Computational Aeroscience. In response to these challenges, the Program is looking toward new massively parallel systems and AI, and is aggressively pursuing a research program in advanced architectures and algorithms and scientific computing environments.

References

[1] Chapman, D. R., Mark, H., and Pirtle, M. W., "Computers vs. Wind Tunnels for Aerodynamic Flow Simulations," AIAA Astronautics and Aeronautics, Vol. 13, April 1975, pp. 22-30 and 35.

[2] Peterson, V. L., and Ballhaus, W. F., Jr, "History of the Numerical Aerodynamic Simulation Program," NASA CP 2454, March 1987.

[3] Blaylock, B. T., and Bailey, F. R., "Status and Future Developments of the NAS Processing System Network," Proceedings of the Third International Conference on Supercomputing, Boston, Mass., May 15-20, 1988.

[4] Choi, D., and Levit, C., "An Implementation of a Distributed Interactive Graphics System for a Supercomputer Environment," International Journal of Supercomputing Applications, Vol. 1, No. 3, Winter 1987, pp. 82-95.

[5] Rogers, S. E., Buning, P. C., and Merritt, F. J., "Distributed Interactive Graphics Applications in Computational Fluid Dynamics," International Journal of Supercomputing Applications, Vol. 1, No.3, Winter 1987, pp. 96-105.

[6] Lasinski, T., Buning, P., Choi, D., Rogers, S., Bancroft, G., and Merritt, F., "Flow Visualization of CFD Using Graphics Workstations," AIAA Paper 87-1180-CP, 1987.

[7] Watson, V., Buning, P., Choi, D., Bancroft, G., Merritt, F., and Rogers, S., "Use of Computer Graphics for Visualization of Flow Fields," AIAA Aerospace Engineering Conference and Show, Los Angeles, CA, February 17-19, 1987.

[8] Fujii, K., Van Dalsem, W. R., Schiff, L. B., and Steger, J. L., "Numerical Simulation over a Strake-Delta Wing, " AIAA Paper 87-1229, 1987.

[9] Robinson, S. K., and Kline, S. J., "Spatial Character and Time-Evolution of Coherent Structures in a Numerically Simulated Boundary Layer," First National Fluid Dynamics Congress, Cincinnati, Ohio, July, 1988.

[10] Rao, K. V. and Van Dalsem, W. R., "Downwash Effects of Multiple Jets on a Delta Wing in Ground Effect," in preparation.

[11] Degani, D. and Zilliac, G. G., "Experimental Study of Unsteadiness of the Flow Around an Ogive Cylinder at Incidence," to be presented at the AIAA Atmospheric and Flight Sciences Conference, Paper No. 88-4330-CP, August 17, 1988.

[12] Bachalo, W. D., and Johnson, D. A., "An Investigation of Transonic Turbulent Boundary Layer Separation Generated on an Axisymmetric Flow Model," AIAA Paper 79-1479, 1979.

[13] Lawrence, S. L., and Jettmar, R. U., "Hypersonic Flow Field Prediction Using Space Marching and Time Dependent Upwind Codes," in preparation.

[14] Kwak, D., Chang, D. L. C., Shanks, S. P., and Chakravarthy, S., "A Three-Dimensional Incompressible Navier-Stokes Flow Solver Using Primative Variables," AIAA Journal, Vol. 24, No. 3, Mar. 1986, pp. 390-396.

[15] Buning, P., Chiu, I. Obayashi, S., Rizk, Y., and Steger, J. L., "Numerical Simulation of the Integrated Space Shuttle Vehicle in Ascent," AIAA Atmospheric Flight Mechanics Meeting, Minneapolis, Minnesota, August, 1988.

[16] Kutler, P., "A Perspective of Theoretical and Applied Computational Fluid Dynamics," AIAA Journal, Vol. 23, No. 3, March, 1985, pp. 328-341.

[17] Dwoyer, D. L., Kutler, P., and Povinelli, L. A., "Retooling CFD for Hypersonic Aircraft," Aerospace America, Vol. 10, October, 1987.

[18] Kutler, P., Steger, J. L., and Bailey, F. R., "Status of Computational Fluid Dynamics in the United States, AIAA-87-1135-CP, June, 1987.

[19] Karin, S., and Smith, N. P., The Supercomputer Era, Harcourt Brace Jovanovich, Publishers, New York, 1987.

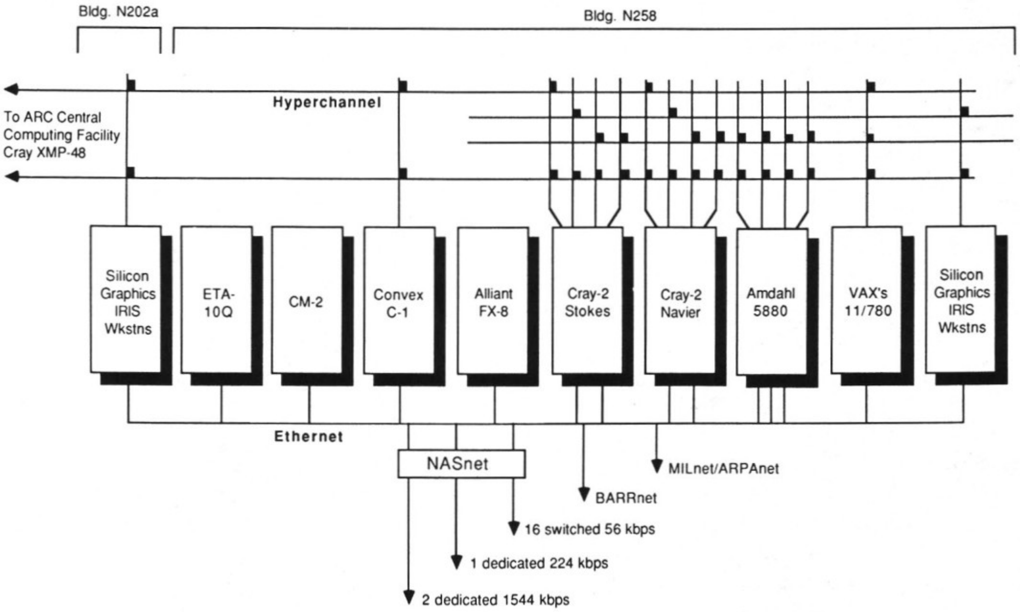


Figure 1. NAS Processing System Network as of April, 1988.

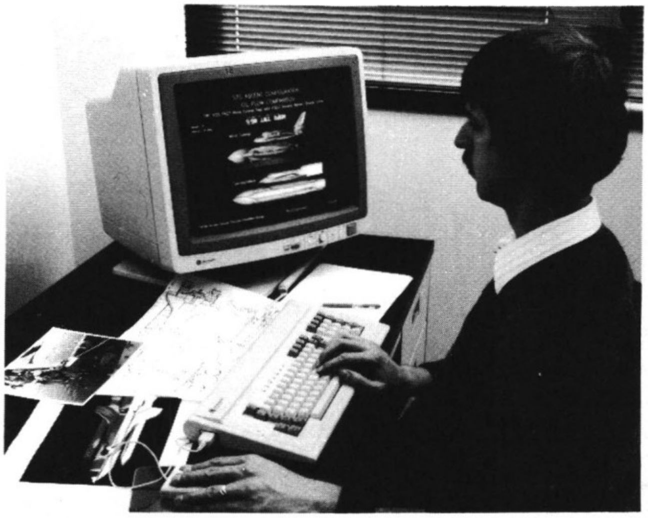


Figure 2. Typical IRIS Workstation.

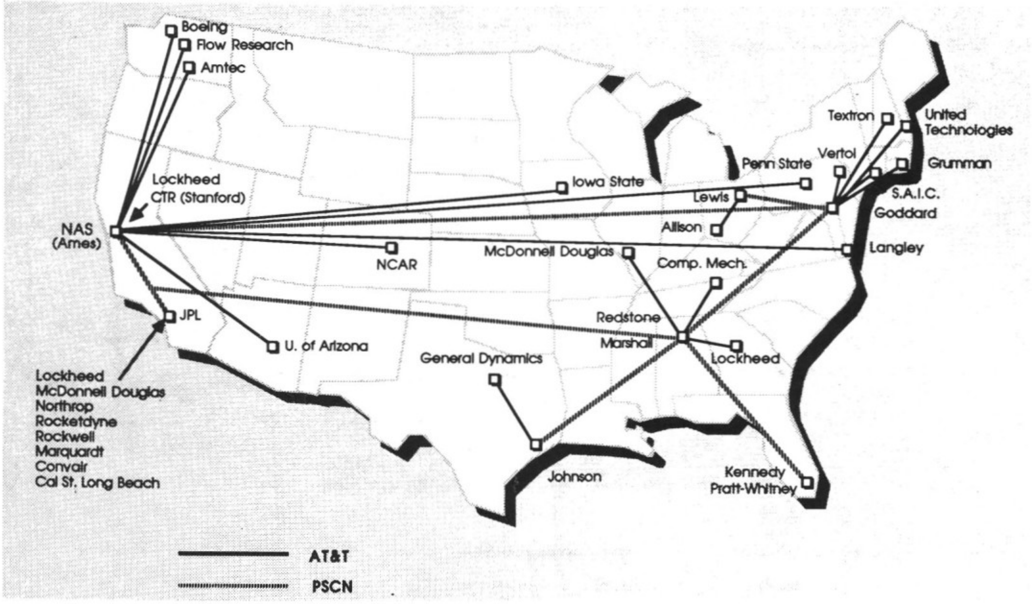


Figure 3. NASnet Communication Links.

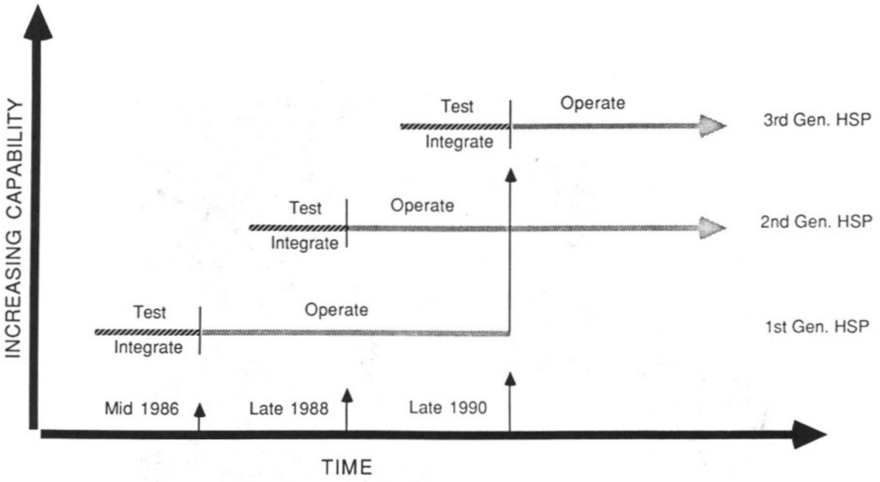


Figure 4. NAS Implementation Strategy.

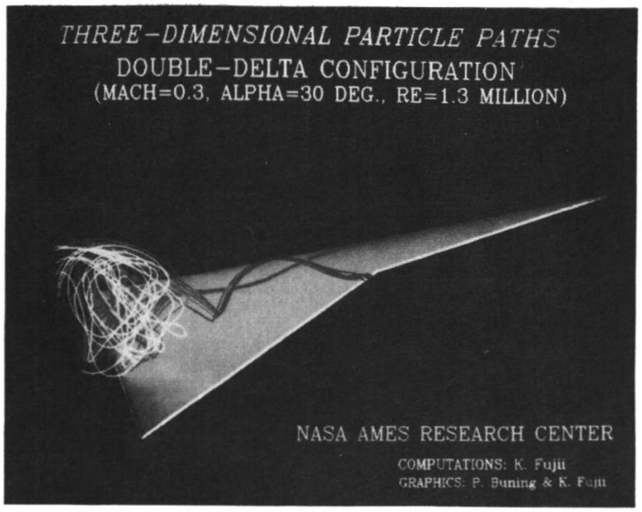


Figure 5. Particle Traces Over a Strake-Delta Wing Demonstrating Vortex Breakdown.

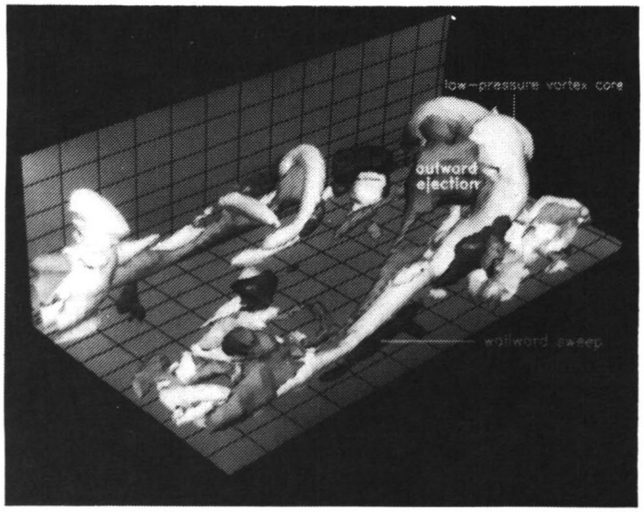


Figure 6. Numerical Simulation of a Turbulent Boundary Layer on a Flat Plate.

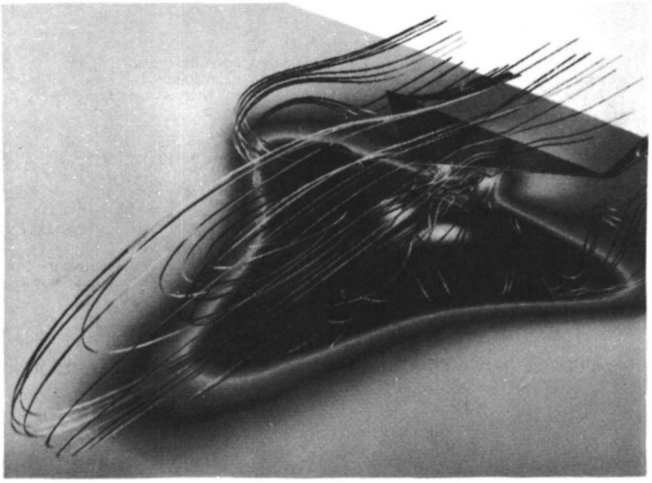


Figure 7. Particle Traces Generated by Jets Emanating from a Delta Wing in a Cross Flow.

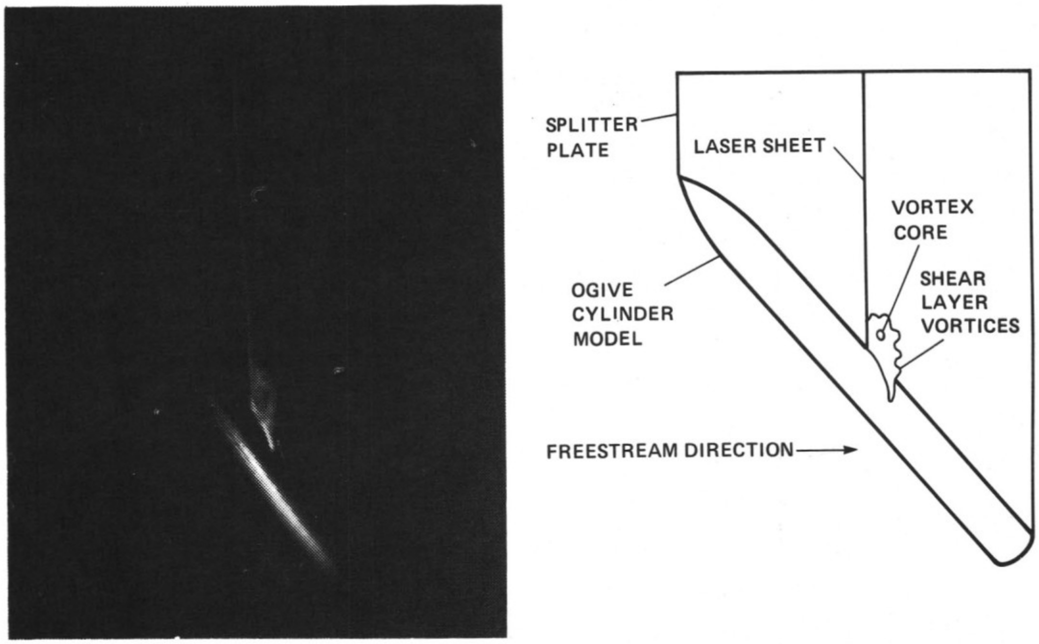
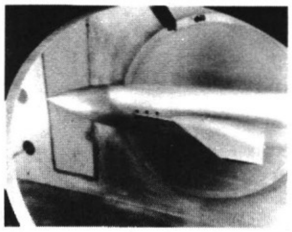


Figure 8. Smoke Laser Sheet on Leeside of Ogive Cylinder at Large Angle of Attack.

MODEL INSTALLATION



MODEL
  BODY: OGIVE CYLINDER
  WING: NACA 64A008
    L.E. SWEEP ANGLE = 37°
    $\mathcal{R} = 3$

TEST CONDITIONS
  SOLID TUNNEL WALLS
  M_∞ = 0.5 TO 0.82
  $Re_c = 1 \times 10^6$ TO 10×10^6
  α = 0 TO 15°

MEASUREMENTS
  MODEL: p, OIL FLOW
  TUNNEL WALLS: p, OIL FLOW, BOUNDARY LAYER THICKNESS (MOUNTING WALL)
  FLOW FIELD: u, v USING LDA

Figure 9. Transonic Wing-Body Combination Mounted in the NASA Ames High Reynolds Number Facility.

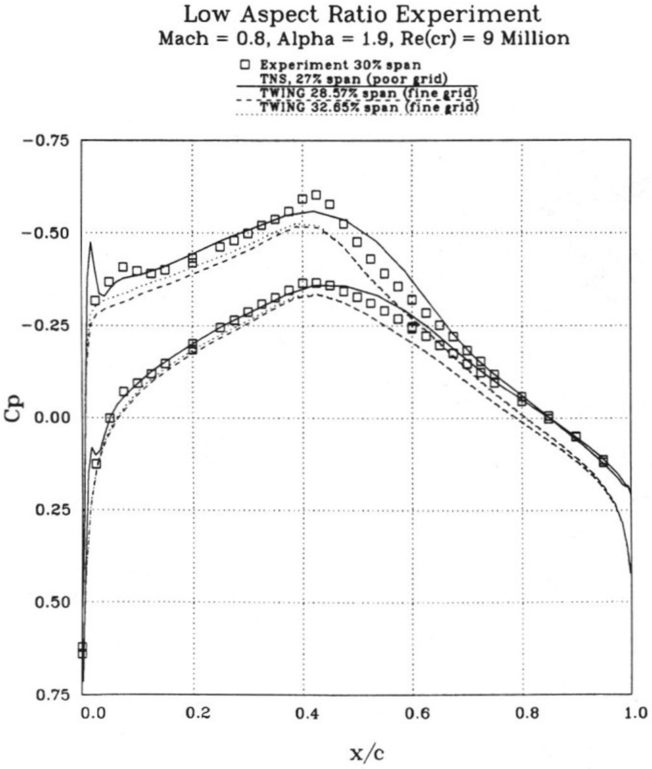
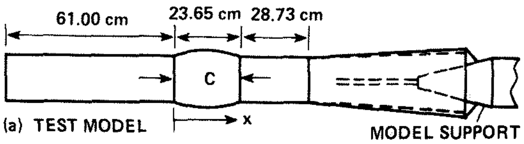
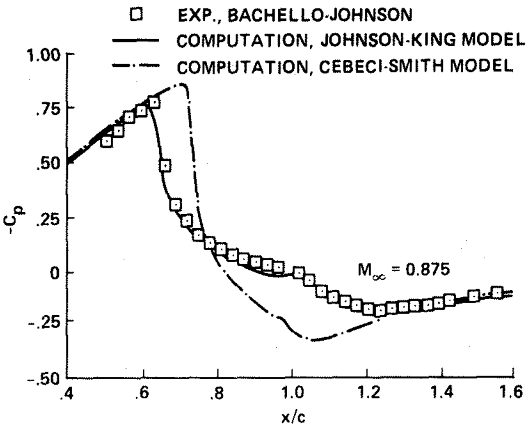


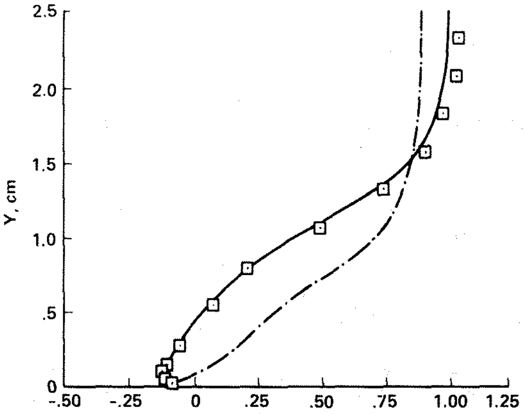
Figure 10. Surface-Pressure Distribution for Low-Aspect Ratio Wing (M = 0.8, alpha = 1.9, Re= 9 M).



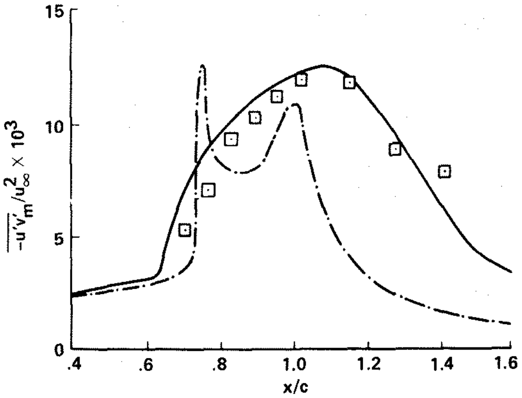
(a) TEST MODEL



(b) PRESSURE COEFFICIENT



(c) VELOCITY, x/c = 1.0



(d) MAXIMUM TURBULENT SHEAR STRESS

Figure 11. Shock Wave, Boundary Layer Interaction.

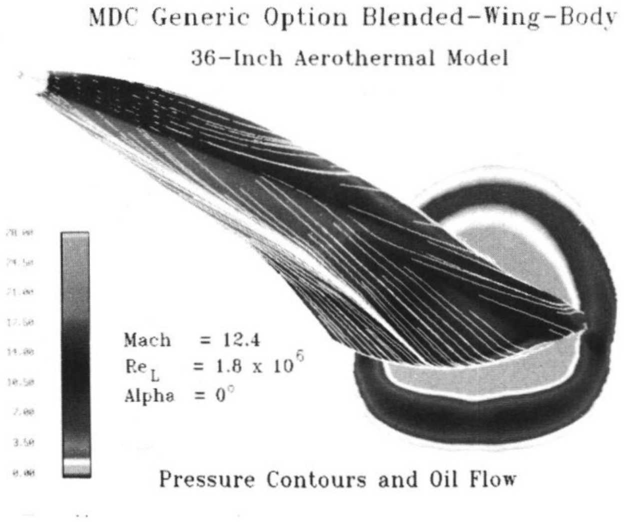


Figure 12. Surface Pressure Contours and Particle Streamlines for Generic Hypersonic Vehicle.

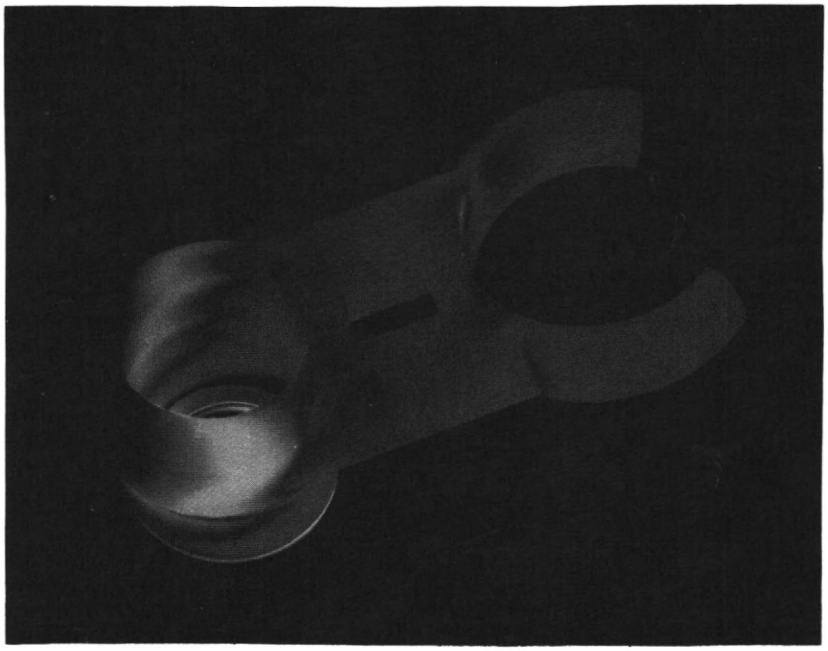


Figure 13. Pressure Map of Two-Transfer Duct Design of the Space Shuttle Main Engine Powerhead.

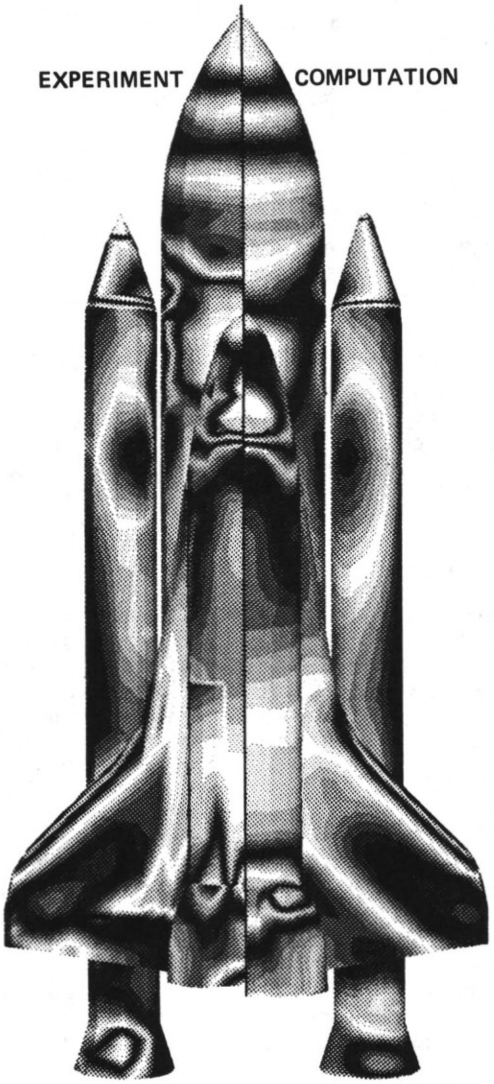


Figure 14. Computed and Experimental Surface-Pressure Data for the Space Shuttle, External Tank, and Solid-Rocket Boosters.