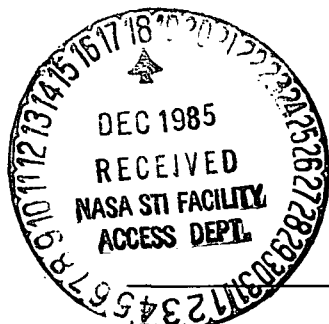


NASA Technical Memorandum 86769



Symposium on Vector and Parallel Processors for Scientific Computation

Victor L. Peterson

(NASA-TM-86769). COMPUTATIONAL FLUID
DYNAMICS AT NASA AMES AND THE NUMERICAL
AERODYNAMIC SIMULATION PROGRAM (NASA) 23 p
HC A02/MF A01 CSCL 20D

N86-15626

Unclas
G3/34 04987

September 1985



National Aeronautics and
Space Administration

Symposium on Vector and Parallel Processors for Scientific Computation

Victor L. Peterson, Ames Research Center, Moffett Field, California

September 1985



National Aeronautics and
Space Administration

Ames Research Center
Moffett Field, California 94035

SUMMARY

Computers are playing an increasingly important role in the field of aerodynamics such that they now serve as a major complement to wind tunnels in aerospace research and development. Factors pacing advances in computational aerodynamics are identified, including the amount of computational power required to take the next major step in the discipline. The four main areas of computational aerodynamics research at NASA Ames Research Center which are directed toward extending the state of the art are identified and discussed. Example results obtained from approximate forms of the governing equations are presented and discussed, both in the context of levels of computer power required and the degree to which they either further the frontiers of research or apply to programs of practical importance. Finally, the Numerical Aerodynamic Simulation Program--with its 1988 target of achieving a sustained computational rate of 1 billion floating-point operations per second--is discussed in terms of its goals, status, and its projected effect on the future of computational aerodynamics.

INTRODUCTION

Computational fluid dynamics has emerged in the last decade as a powerful new tool for research and development associated with all types of aerospace vehicles. The reasons are manifold; although they generally relate to the increasing availability of very powerful supercomputers, the improving methodology which makes it easier to solve more complete forms of the governing equations numerically, and a growing number of examples where computers have enabled substantial design improvements to be made rapidly and cost effectively.

The rate of development of computational fluid dynamics, although rapid, is being limited by the rate of improvements in computer power (speed and memory). The amount of computer power required for computational fluid dynamics depends on three factors: 1) the complexity of the problem, which includes the fluid physics governing the problem to be solved and the complexity of the geometry about which the fluid moves; 2) the efficiency of the algorithm available for solving the governing equations numerically; and 3) the amount of computation time that can be invested to solve the problem at hand. The power of the largest currently available supercomputers (Class VI machines), such as the CYBER-205 and the Cray-XMP, is far less than that required to solve the most complex problems involving the full Navier-Stokes equations and realistic aircraft shapes in a practical amount of computation time, even though these machines can sustain computational rates greater than 100 million

floating-point operations per second (Mflops) and can have electronic main memories in excess of 10 million words. Thus, the computational fluid dynamicist is limited by available computer power and must resort to making approximations either to the physics, to the geometry, or to both to obtain problem solutions. A new generation of supercomputers, now under development and expected to be available within the next year or two, will remove some of the limitations and enable another surge in the advancement of the discipline.

The objectives of this paper are to provide a brief review of the factors motivating the development of computational fluid dynamics, an outline of the program at NASA Ames Research Center aimed at advancing the discipline, a discussion of several example applications, and a description of the Numerical Aerodynamic Simulation (NAS) program which has been undertaken to provide the U.S. aerospace community with a new national capability.

MOTIVATIONS, EQUATIONS, AND COMPUTER REQUIREMENTS

Aerodynamics research and development was based largely on experimentation and linearized inviscid theory until about a decade ago when digital computers and numerical methodology reached the necessary degree of maturity to solve for flows about realistically shaped aircraft components using nonlinear forms of the governing equations. Since that time, numerical solutions have provided an increasingly important complement to, and sometimes a substitute for, experiments conducted in wind tunnels and in flight. Today, the two principal tools of the aerodynamicist are wind tunnels and computers, and the balance between the use of these tools is beginning to shift in the direction of the computer.

Both technical and economic factors are driving the aerodynamicists toward computers, even though achievements in aeronautics continue to be made largely on the basis of experiment. The inherent limitations of wind tunnels as the primary tool for developing and verifying the design of new aerospace vehicles have become increasingly apparent as aircraft have become more complex and flight envelopes are extended. For example, the walls in wind tunnels and the supports used to hold the models severely restrict the accuracy of measurements obtained at near-sonic speeds. Aeroelastic distortions, which are known to be always present in flight, cannot be accurately simulated with subscale models tested in wind tunnels. Furthermore, limitations on temperature and on the working fluid restrict the ability of the wind tunnel to simulate the flight of vehicles reentering Earth's atmosphere from space and the entry of probes into other planetary atmospheres. These limitations did not seriously affect the usefulness of wind tunnels during the era of simpler, low-speed flight. On the one hand, the increased complexity and broadened performance envelopes of aircraft have increased the number of wind tunnel hours expended in the development of new aircraft, and the cost per hour of testing has been increasing substantially at the same time. On the other hand, the cost of numerically simulating a given flow is decreasing at a rapid rate because of improvements in algorithms and computer cost effectiveness, and the accuracy of

computer simulations is improving as the computers are becoming more powerful. Therefore, there are compelling reasons to advance the state of the art in computational aerodynamics.

The goal of computational aerodynamics is to obtain, by numerical simulation, aerodynamic information normally measured in wind tunnels, flight tests, or other ground-based facilities. The major benefits will be significantly improved designs resulting from the use of numerical optimization procedures and the simulation of full-scale, free-flight conditions; increased efficiency of wind-tunnel testing; and reduced time, cost, and risk in the development of aerospace vehicles.

The amount of computer power required to advance the capability of computational fluid dynamics to treat increasingly complex aerodynamic flows involving ideal gases is indicated by the data in table I. Solutions to increasingly refined approximations to the full Navier-Stokes equations provide additional information of value to the designer, but they also require increasing refinements to the computational mesh to capture the additional detail in the flow physics. This requirement places an increasing burden on the computer used to solve the equations. The computer requirements given in the table are based on obtaining solutions, using current algorithms, to three-dimensional problems involving the number of grid points shown in 10 to 15 min. Class VI computers are adequate for the inviscid-flow technology, but they fall far short of meeting the requirements for problems dominated by strongly coupled viscous effects. The next major step forward for the discipline of computational aerodynamics can be taken when computers about 30 times more powerful than the Class VI machines become available. Computers with this amount of power will, for the first time, permit the Reynolds-averaged form of the Navier-Stokes equations to be routinely used for calculating flows about relatively complete aircraft geometries. A more complete discussion of the levels of approximation to the governing equations can be found in reference 1.

COMPUTATIONAL FLUID DYNAMICS PROGRAM AT AMES RESEARCH CENTER

Theoretical fluid dynamics and aerodynamics have been subjects of research at Ames Research Center ever since the laboratory was founded in 1940 as part of NASA's predecessor, the National Advisory Committee for Aeronautics (NACA). Prior to 1955, this research was done largely through closed-form analytical analysis, although some computational work was done using desk-top mechanical calculators. Of course, the theoretical work was complemented by considerable experimentation. The first electronic digital computer for scientific work at Ames Research Center, an IBM 650, was installed in 1955. Over the next 14 years, successively more powerful machines such as the IBM 704, 7090, and 7094 were brought into use. Concurrently, there was a gradual migration of the research efforts from the desk-top calculators to the electronic digital computers. The beginning of the focused thrust in computational fluid dynamics occurred in about 1969 with the installation of the IBM 360/67. This machine was the first to provide an interactive computing environment, a feature that was particularly good for stimulating the numerical experimentation required

for rapid progress in algorithm development. The program at Ames Research Center has continued to expand over the years along with the power of scientific computers. Currently, two supercomputers, a Cray-XMP and a CYBER-205, are being used in the computational program.

The technical scope of the Ames Research Center's program in computational fluid dynamics is outlined in table II. The four main areas of work are: 1) basic research, 2) applied research, 3) applications, and 4) validation. Each of these areas is discussed briefly in the remainder of this section.

Basic Research

The most fundamental work involved investigations of transition from laminar to turbulent flow and the physics of turbulence. Much of this work requires solving the complete Navier-Stokes equations, although some of it is being done using the Large-Eddy Simulation form of the equations. Therefore, problems are limited to simple flow situations that can be treated with computational domains involving no more than about 17 million nodes (256^3). Some of the problems just being undertaken are boundary-layer transition on a flat plate, flat plate turbulent boundary layers in adverse pressure gradients, and the separating and reattaching turbulent flow over a rearward-facing step. The scope of this work is limited by the power of available supercomputers since a single case for each of the problems mentioned requires about 100 hr of processing time on today's machines. This basic work is aimed at finding methods for solving practical problems such as controlling transition in aerodynamic flows, enhancing mixing rates, reducing aerodynamic drag, and improving turbulence models for use in the Reynolds-averaged form of the equations.

Applied Research

The development of computer codes and their application to practical problems in fluid dynamics and aerodynamics requires the consideration of several different contributing technologies. Some of these technologies (such as the numerical representation of geometric shapes and the computational grids surrounding them, and numerical algorithms that describe the governing equations in a form that can be solved by the computer) are required in all situations. Other technologies (such as the modeling of turbulence and the description of the chemistry of reacting gases) are required only in cases where these physical processes are present. Still other technologies (such as numerical optimization, techniques for graphical display of numerical results, and the use of expert systems) are optional but they serve to enhance the computational process. Research is under way on all of these contributing technologies so that the greatest possible advances in the discipline can be made.

The work in algorithm development has been particularly effective. In fact, reductions in the effective cost of performing a computation because of improvements in computer speed over the past 20 years have been closely paralleled by reductions made possible by improvements in numerical methods. Solution time has been reduced

by several orders of magnitude as a result of this research and at least two more orders of magnitude appears to be within the realm of possibility.

The treatment of increasingly complex aerodynamic configurations and flow physics requires that more attention be placed on the development of methods for generating computational grids. The quality of computed results is closely tied to the proper placement of the points in the flow where the solution to the flow equations is sought. Points must be closely spaced in regions of high-flow gradients, and yet the total number of grid points used in a problem must be kept as small as possible to obtain problem solutions in a reasonable amount of time. The locations of high gradients usually are not known a priori, particularly in time-dependent flows, so that work must continue on the improvement of solution-adaptive grids. Another requirement is that the grids must conform closely to the shape of the body under study, at least in regions close to the body surface. When body shape is changing with time or when multiple bodies moving relative to each other are being studied, then component-adaptive grids may be required. This set of conditions can result in the requirement to interface multiple grid systems moving relative to each other. Maintaining numerical stability of algorithms when grids are moving with time and assuring the continuity of flow variables across moving boundaries provides interesting challenges to the researcher.

Calculation of aircraft performance requires the consideration of viscous effects and many of the problems of engineering importance involve turbulent flow. All of the physics of turbulence are contained in the Navier-Stokes equations, but these equations usually cannot be solved in their complete form because of the wide range of scales of fluid motions that must be resolved. Therefore, considerable attention is being placed on solving the Reynolds-averaged form of the equations in which terms accounting for the time-averaged transport of momentum and energy must be modeled. The turbulence models depend on constants which until recently could be obtained only through physical experiments. It is relatively easy to find a set of constants that will apply to one particular fluid flow, but it is difficult to develop a single model that will apply universally to all flows. Current models are generally adequate for problems involving only small regions of flow separation, but improvements are being sought to handle the more general situations.

Problems involving the high-speed motion of bodies and those problems involving combustion require the consideration of chemically reacting real gases. Additional equations to account for the physical processes add further complexity to the numerical treatment of the problems. This places greater demands on computer speed and memory as well as solution algorithms, grids, and turbulence models. Research is under way to provide the necessary advances required to solve these problems.

Computational methods offer the promise of greatly enhancing the process of optimizing the design of an aerospace vehicle. Empirical approaches are inefficient, even when attention is limited to the consideration of a single discipline such as aerodynamics. Integrating multiple disciplines such as aerodynamics, propulsion, structures, and controls is even more complicated. Powerful numerical optimization techniques eventually will provide a new tool for rapidly and systematically designing a vehicle with superior performance since all of the necessary

design compromises and constraints will be met in an optimum fashion. The power of currently available computers is sufficient to treat individual aircraft components within a single discipline such as the aerodynamic or structural optimization of a wing. However, more powerful computers and considerable research on optimization methods are required to meet the long-term goals.

The solution of three-dimensional problems on supercomputers involves the management of enormous amounts of output data. The amount of output data presents a significant problem to the scientist or engineer. The only practical way to make use of much of the data is through the use of graphical display devices to permit the visualization of the flow phenomena. These devices and some supporting software are available, but much remains to be done to improve their functionality. Research is under way to simplify the process of displaying multicolor three-dimensional images of the data and to record the information in motion-picture form.

Finally, a new area of research focus involving the application of expert-systems concepts is beginning to emerge. The goal is to automate more of the computational process to make more effective use of an investigator's time and to put the codes into the hands of a broader range of users. Many steps associated with the computational process such as geometry modeling, grid generation, and post processing of results are labor-intensive and must be repeated each time a new problem is undertaken. Most of these tasks, as well as the selection of suitable turbulence models, currently requires the attention of experts having considerable training. Widespread use of computer codes in a design environment will require the entire process to be simplified and made more efficient. Initial efforts are now being directed toward the simplification of the grid-generation process and the development of an overall framework for an automated-design code.

Applications and Validation

The ultimate goal of computational fluid dynamics is to solve problems of practical scientific or engineering importance. Therefore, considerable effort is being placed on integrating the various contributing technologies into pilot codes which can be tested on problems representative of those of interest in aerospace research and development. Codes for a wide variety of example problems associated with airframes, rotors, engines, and atmosphere entry vehicles are being developed and tested against experiment. The process of making trial applications is a crucial part of the research in computational fluid dynamics since it is the only way that deficiencies can be identified. Special experiments often are required to validate the computational methods. It is usually not sufficient to test the codes against integrated information such as forces and moments. Instead, detailed pressure distributions are almost always required and other information such as local skin friction and flow-field velocities is highly desired. Care must also be taken to match all important features of the experiment in the computations. In addition to Mach number, Reynolds number, and model attitude, it sometimes is necessary to include in the computations model supports, wind tunnel walls, freestream turbulence levels, and pressure level measured downstream of the model in the wind tunnel

diffuser if comparisons are being made with measurements made in wind tunnels. After the pilot codes are thoroughly tested and their ranges of applicability clearly understood, they can be integrated into the design environment. Examples of several applications of pilot codes to problems of interest are presented in the next section.

EXAMPLE APPLICATIONS

The ultimate goal of computational fluid dynamics is to solve problems of practical scientific or engineering importance. Since the discipline is not yet mature, it is necessary to select problems that are suitable for solution with available methodology and computers. One of the efforts at Ames Research Center is to develop codes that will test the state of the art in problem solving and apply them to situations of contemporary interest. Results of three such pilot applications are presented and discussed.

Space Shuttle Orbiter

Flow fields about the Space Shuttle Orbiter have been calculated under the assumptions of perfect gas and fully turbulent flow. These calculations were made using a combination of two different codes: one is based on the full, Reynolds-averaged, Navier-Stokes equations, and the other is based on the "parabolized" Navier-Stokes equations in which time-dependent terms and streamwise derivatives of viscous-flow terms are neglected. The first code was used in regions where the flow is either entirely subsonic or mixed subsonic and supersonic; the second code was used in regions of entirely supersonic flow. Details of the procedure are described in reference 2.

Sample results for one particular point in the atmosphere entry trajectory are presented in figure 1. Conditions chosen for this example are a Mach number of 7.9, angle of attack of 25° , wall temperature of 300 K, and a unit Reynolds number of about $2.4 \times 10^6/\text{m}$. These results are superimposed on a computer-generated image of the orbiter which was displayed on a graphics monitor. Lines on the surface of the vehicle represent the directions of the shear-flow vectors. These directions compare favorably with results of wind tunnel tests in which oil-coated models were photographed after being subjected to comparable test conditions. In addition to local flow directions, results of this type provide the locations of lines of flow separation and reattachment. The lines that are not directly on the surface represent computer-generated particle paths. These particle paths give additional information about the flow such as the origins and paths of vortices. Other information that can be derived from the calculations includes aerodynamic forces and moments, surface pressures, heat transfer, temperatures, and all of the other flow variables of interest.

Space Shuttle Main-Engine Analyses

A computer code is under development to analyze the internal flow in the Space Shuttle main-engine (SSME) hot-gas manifold. The SSME powerhead component arrangement is shown in figure 2. Hot gas flows from the fuel preburner down to the inner annular duct, through a gas turbine, turns around in a 180° bend, and goes into the outer, annular fuel bowl. Then it flows through three transfer tubes and enters into the region of the main-injector posts. Around the bottom portion of each post there are a number of small holes through which the hot gas flows into the main combustion chamber. There it mixes with the oxidizer, which comes through a small circular passage along the centerline of each post. The Reynolds number of the primary flow in the manifold is about $39 \times 10^6/m$. Because of the extremely high gas temperature, the Mach number is less than 0.12. The flow is turbulent and practically incompressible. The goal of this work is to develop a code that can be used in the design of an engine with substantially increased operating margin and durability. Details of this work are presented in reference 3.

Initial results have been obtained for the laminar flow through the hot-gas-manifold configuration shown by the computer-generated image of the geometry presented in figure 3. Lines on the surfaces represent the computational mesh. Altogether, about 90,000 grid points were used in this analysis. Computed velocity vectors for the flow through the turnaround duct, fuel bowl, and center transfer tube are displayed in figure 4. The flow throughout is highly nonuniform. Large regions of flow separation are formed along the inner wall downstream of the 180° bend and just downstream of the entrance to the transfer tube. Results of this work, validated by experiment, have shown that the center transfer tube carries only about 9.1% of the flow. These results suggest that a more efficient design can be found by changing the contours of the inner passages and by eliminating the center transfer tube.

Measured pressure losses in the original hot-gas manifold are compared with those in two computationally improved designs in figure 5. These results show that it is, indeed, possible to improve the flow through the engine. Furthermore, the measured and computed results compare very favorably in spite of the many approximations made in the computations. This example clearly shows how computations complement experiment by providing diagnostic information that is difficult, is not impossible, to obtain by any other means. The next step, now in progress, is to extend the computations to include the effects of turbulence.

Flow through Rotor-Stator Passages of an Axial Turbine

Another computer code under development treats the unsteady flow through rows of rotor-stator blades in a turbine where rotor blades are moving relative to fixed stator blades. The geometric complexity is such that it is only practical to treat this type of flow in two dimensions with currently available computers. Calculations based on the Euler equations have been performed for blades having parabolic-arc airfoils immersed in a supersonic freestream. The rotor blades were forced to

move relative to the stator blades at a Mach number of 0.1 based on free-stream conditions. Details of this work are given in reference 4.

Calculated pressure contours at one instant of time are shown in figure 6. Downward motion of the forward airfoils, and the flows associated with them, creates changing upstream conditions on the fixed aft airfoils. The resulting complex, time-dependent, interacting pressure patterns would be extremely difficult, if not impossible, to obtain from experiment. Of course, all of the flow quantities of interest are provided by the computations, including the fluctuating loads on the airfoils. Motion pictures of these types of unsteady flows are particularly good for displaying the time-dependent phenomena. The next step in this work is to solve the Reynolds-averaged, time-dependent, Navier-Stokes equations using more realistic airfoil shapes and upstream flow conditions.

NUMERICAL AERODYNAMIC SIMULATION PROGRAM

Available computer power is limiting the development of the discipline of computational fluid dynamics. Existing computers are adequate to treat either complex flow geometries with simple physics or simple flow geometries with complex physics, but they cannot handle the combination of complex geometries with complex physics. However, computer technology is advancing rapidly and it seems should be possible to routinely use the Reynolds-averaged form of the Navier-Stokes equations in complex applications, provided the most advanced computers are made available in a timely manner. The NAS program was initiated to meet this need. The NAS program will provide the most advanced computer facility possible to permit computationally oriented disciplines to advance rapidly along with the latest developments in computer equipment. The program received formal approval in October 1983, after nearly 8 yr of focused studies. Equipment is being assembled, and the initial operational capability is scheduled for the summer of 1986.

The principal objectives of the NAS program are twofold: 1) to act as the pathfinder in advanced, large-scale, computer-system capability through systematic incorporation of state-of-the-art improvements in computer hardware and software, and 2) to provide a national computational capability available to NASA, Department of Defense, other government agencies, industry, and universities as a necessary element in ensuring continuing leadership in computational aerodynamics and related disciplines. The key technical goals of the program are summarized as follows.

1. Initial operating capability in 1986

- a. First high-speed processor (HSP-1)
 - Type: Cray-2
 - Speed: 250 Mflops sustained
 - Memory: 256 million, 64-bit, directly addressible words
- b. Integrated and expandable network
- c. Uniform access to all subsystems
- d. Initial, long-haul, communications capability

2. Extended operating capability in 1988
 - a. Second additional high-speed processor (HSP-2)
Speed: four to six times that of HSP-1
Memory: 1 billion, 64-bit, directly addressible words
 - b. Upgraded subsystems and graphics subsystem
 - c. Wide-band communications to national user community
 - d. Department of Defense classified, secure operations
3. Further extensions in early 1990s
 - a. Replacement for HSP-1
Speed: four to six times more powerful than HSP-2
 - b. Upgraded long-haul communications

NAS Processing System Network

The NAS Processing System Network (NPSN) will be a large-scale, distributed network at Ames Research Center. The network will offer facilities to support every aspect of computational aerodynamics, from "number crunching" for interactive, aerodynamic-flow-model solutions to real-time, graphical-output display manipulation. Its equipment will span the performance range from supercomputers to micro-processor-based workstations. The NPSN resources will be made available to a nationwide community of users via interfaces to landline and satellite data communications links.

The NAS program is structured to accommodate the continuous development of the NPSN as a leading-edge, computer-system resource. An important element in this strategy of staged growth will be the ability to accommodate new supercomputers from different vendors with minimum disruption of operational use of the existing network. The NPSN will consist of the following seven subsystems: High-Speed Processor (HSP), Support Processing Subsystem (SPS), Workstations Subsystem (WKS)m, Graphics Subsystem (GRS), Mass Storage Subsystem (MSS), Long-Haul Communications Subsystem (LHCS), and High-Speed Data Network (HSDN). Only the first four will be programmed by users. A sketch of the complete NPSN is shown in figure 7.

High-Speed Processor Subsystem- The HSP is the advanced scientific-computing resource within the NPSN. In addition to doing batch processing, the HSP will do interactive time-sharing processing, which will aid in debugging applications programs, editing results, and performing other activities that depend on close coupling of user and processor to achieve optimum productivity.

Present plans call for two generations of HSP computers to be in the system at any one time. The HSP-1 will process optimally structured computational-aerodynamics applications at a sustained rate of 250 Mflops and have a main-memory capacity of 256-million, 64-bit words. The Cray-2 supercomputer selected for HSP-1 will be installed in the NPSN in September 1985. The Cray-2 has a four-processor configuration with an aggregate peak computing rate of 2,000 Mflops. The HSP-2, planned for installation in 1987, is expected to operate at a sustained rate of 1,000 Mflops and have a main memory ranging to 1-billion, 64-bit words.

Support-Processing Subsystem- The SPS main-frame computer does general-purpose interactive processing for local and remote terminal-based users (those without workstations) and has intermediate performance between the HSP and WKS. The SPS will have unit record input/output devices (high-speed printers and microfilm) and will be the focal point for network monitoring and system operation. Two machines will be installed to handle the initial workload and to provide redundancy.

Workstation Subsystem- While HSP serves the global user community, WKS, a microprocessor-based resource for the individual researcher, will serve as a "scientist's workbench" to perform text and data editing, to process and view graphics files, and to perform small-scale processing. Each individual workstation will have the appropriate memory, disk storage, and hard-copy resources to fit the local user's needs. Individual clusters of workstations will be networked through the HSDN for use within local user groups. In addition to local processing, the WKS will give access to other user-programmed systems on the network and allow file transfer.

The IRIS (Integrated Raster Imaging System) 1500 has been selected as the initial workstation. It uses a general-purpose Motorola 68010 microprocessor with 2.5 megabytes (MB) of central memory, a 474-MB disk, high-resolution, multicolor display, and communication interfaces to both Ethernet and Hyperchannel networks. A key element of the IRIS, a set of special-purpose microprocessors called a "geometry engine," transforms and displays three-dimensional flow data at rates exceeding 50,000 points/sec.

Graphics Subsystem- The GRS, a special-purpose system, will produce sophisticated graphical displays of applications calling for highly interactive, high-density graphics for input preparation and result analysis. The GRS will provide levels of performance and storage beyond those provided by workstations and will be shared by many users.

Mass-Storage Subsystem- The MSS will serve the global on-line and archival file-storage needs of the NPSN. This subsystem will check and coordinate requests for files to be stored or retrieved within the NPSN and maintain a directory of all contained data. The MSS will act as a file server for other NPSN subsystems; will control its own internal devices; and will perform file-duplication, media-migration, storage-allocation, accounting, and file-management functions.

Users of the NPSN will create and use files on various subsystems such as HSP, SPS, GRS, or WKS. However, after the user has exited the NPSN, the main repository of these files will be the MSS. This subsystem will hold these large files that will be used as input to, or generated as output from, the largest tasks that will be processed on the HSP and GRS. The MSS will contain user source and object codes, and parameter and data files that are kept for any significant period of time. The MSS will also contain the backup files that are created to improve the probability that long-lasting or high-value files are accessible when needed.

Long-Haul Communications Subsystem- The LHCS will provide the data-communication interface between the NPSN and data-communication links to sites

geographically remote from Ames Research Center. This subsystem will have the necessary hardware/software interfaces; modulation and demodulation devices; and recording, processing, data buffering, and management functions to support data transfers and job control by remote users.

In the sense that the MSS is a back-end resource for the entire NPSN, the LHCS is a front-end resource. It provides for access by remote users to the HSP, SPS, GRS, and WKS, but it is not specifically addressed or programmed by the user. The LHCS processor functions as a data-communications front-end providing store and forward, protocol conversion, and data concentrator service.

High-Speed Data Network Subsystem- The HSDN allows exchange of data and control messages among NPSN subsystems. Major design emphasis will be placed on its ability to support large file transfers among NPSN subsystems. The HSDN will include high-speed (minimum 50 megabits/sec) interface devices and driver-level network software to support NPSN internal data communications.

NPSN Software

The NPSN will embrace a rich set of systems and utility software aimed at letting users make best use of time. The objective is to provide software that will give common user interfaces across the NPSN, enabling easy movement from subsystem to subsystem, the applications and systems software to be written in high-level portable languages, the reduction of hardware dependence, and the minimization of impedance from network visibility.

To meet this objective, the NPSN will use AT&T's UNIX System V operating system as the common software foundation on each of the user-visible subsystems. Networking software modeled after the 4.2 Berkeley software distribution (bsd) UNIX operating system will be added. (The 4.2 bsd UNIX is a research version of the operating system developed at the University of California, Berkeley, which supports interconnection of heterogeneous mini- and microcomputers using a variety of communications protocols.) The communications protocols selected for the NPSN come from the DOD Internet family of protocols, including transmission control protocol, internet protocol, and datagram protocol.

By offering a common UNIX operating system and basic interprocesses communications, the NPSN will have a software environment capable of supporting existing and new distributed applications, including text editors, graphics display packages, and fluid-dynamics solution codes.

CONCLUDING REMARKS

Computational fluid dynamics is emerging as a powerful new tool in aerospace research and development. Applications in the field of aerodynamics now are widespread and there are many examples of improved aircraft designs made possible by

this new tool. Available supercomputers are not yet powerful enough, however, to solve the full governing equations, except for special cases involving simplified flow geometries and moderate Reynolds numbers. Example applications of current technology to the Space Shuttle Orbiter and to the flowthrough rotor-stator blade passages in an axial turbine, however, show the value of computations in providing information about realistic design situations that would be difficult, if not impossible, to measure. A vigorous research program in computational fluid dynamics has been under way for the past 16 yr at the Ames Research Center. This program is aimed at developing the discipline such that flows about increasingly more complex geometries can be treated with greater degrees of physical precision. More recently, another program has been undertaken to develop the most advanced computational facility possible to assemble for both local and remote use by the U.S. aerospace community. The 1988 goal of this effort, the NAS program, will provide the capability to solve the Reynolds-averaged form of the Navier-Stokes equations about realistic aircraft shapes. This program will be made possible through the incorporation of a high-speed processor having a very large main memory and capable of performing 1-billion floating-point operations/sec. This new facility should serve to promote the advancement of computational fluid dynamics through the remainder of this century and beyond.

REFERENCES

1. Peterson, Victor L.: Impact of Computers on Aerodynamics Research and Development. IEEE Proc., vol. 72, Jan 1984, pp. 68-79.
2. Chaussee, Denny S.; Rizk, Yehia M.; and Buning, Pieter G.: Viscous Computation of a Space Shuttle Flow Field. NASA TM-85977, 1984.
3. Chang, James L.; Kwak, Dochan; Doa, Samuel C.; and Rosen, Robert: A Three-Dimensional Incompressible Flow Simulation Method and its Application to the Space Shuttle Main Engine. Part I - Laminar Flow. AIAA Paper 85-0175, Jan. 1985.
4. Rai, Man Mohan: A Relaxation Approach to Patched-Grid Calculations with the Euler Equations. AIAA Paper 85-0295, Jan. 1985.

TABLE I.- MAJOR LEVELS OF APPROXIMATION TO THE NAVIER-STOKES EQUATIONS WITH RESULTS PROVIDED, AND COMPUTER REQUIREMENTS TO OBTAIN SOLUTIONS IN 15 min OF COMPUTER TIME

APPROXIMATION	CAPABILITY	GRID POINTS REQUIRED	COMPUTER REQUIREMENT
LINEARIZED INVISCID	SUBSONIC/SUPERSONIC PRESSURE LOADS VORTEX DRAG	3×10^3 PANELS	1/10 CLASS VI
NONLINEAR INVISCID	ABOVE PLUS: TRANSONIC PRESSURE LOADS WAVE DRAG	10^5	CLASS VI
REYNOLDS AVERAGED NAVIER-STOKES	ABOVE PLUS: SEPARATION/REATTACHMENT STALL/BUFFET/FLUTTER TOTAL DRAG	10^7	30 x CLASS VI
LARGE EDDY SIMULATION	ABOVE PLUS: TURBULENCE STRUCTURE AERODYNAMIC NOISE	10^9	3000 x CLASS VI
FULL NAVIER-STOKES	ABOVE PLUS: LAMINAR/TURBULENT TRANSITION TURBULENCE DISSIPATION	10^{12} TO 10^{15}	3 MILLION TO 3 BILLION CLASS VI

TABLE II.- TECHNICAL SCOPE OF COMPUTATIONAL AERODYNAMICS PROGRAM AT
AMES RESEARCH CENTER

- BASIC RESEARCH
 - TRANSITION
 - PHYSICS OF TURBULENCE
- APPLIED RESEARCH
 - NUMERICAL ALGORITHMS, BOUNDARY CONDITIONS
 - COMPLEX GEOMETRY REPRESENTATION, GRID GENERATION
 - CHEMICALLY REACTING REAL GASES
 - TURBULENCE MODELS
 - EXPERT SYSTEMS CONCEPTS
 - INTEGRATION OF TECHNOLOGIES INTO PILOT CODES
 - NUMERICAL OPTIMIZATION
 - GRAPHICS
- APPLICATIONS
 - AIRFRAMES, ROTORS, ENGINES, ENTRY VEHICLES
- VALIDATION
 - SELECTED BENCHMARK EXPERIMENTS

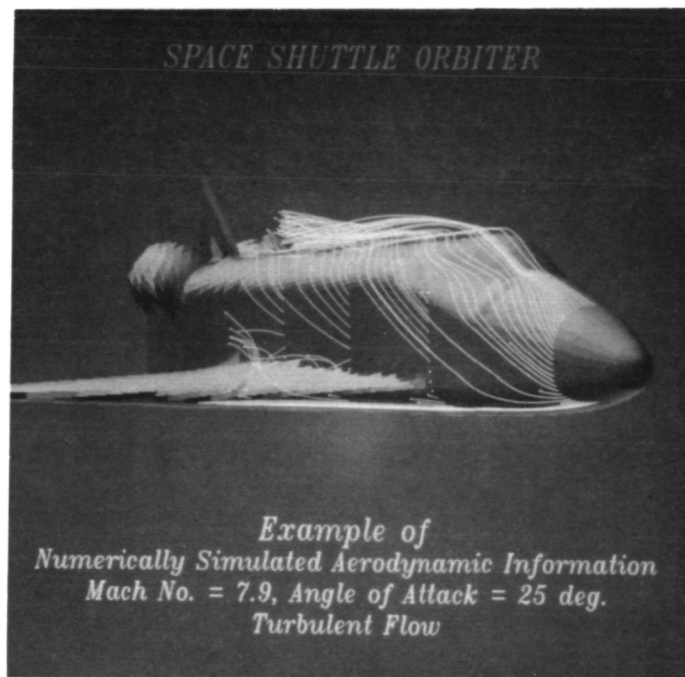


Figure 1.- Example results of calculated airflow about the Space Shuttle Orbiter; surface shear-flow vectors and computer-generated particle paths.

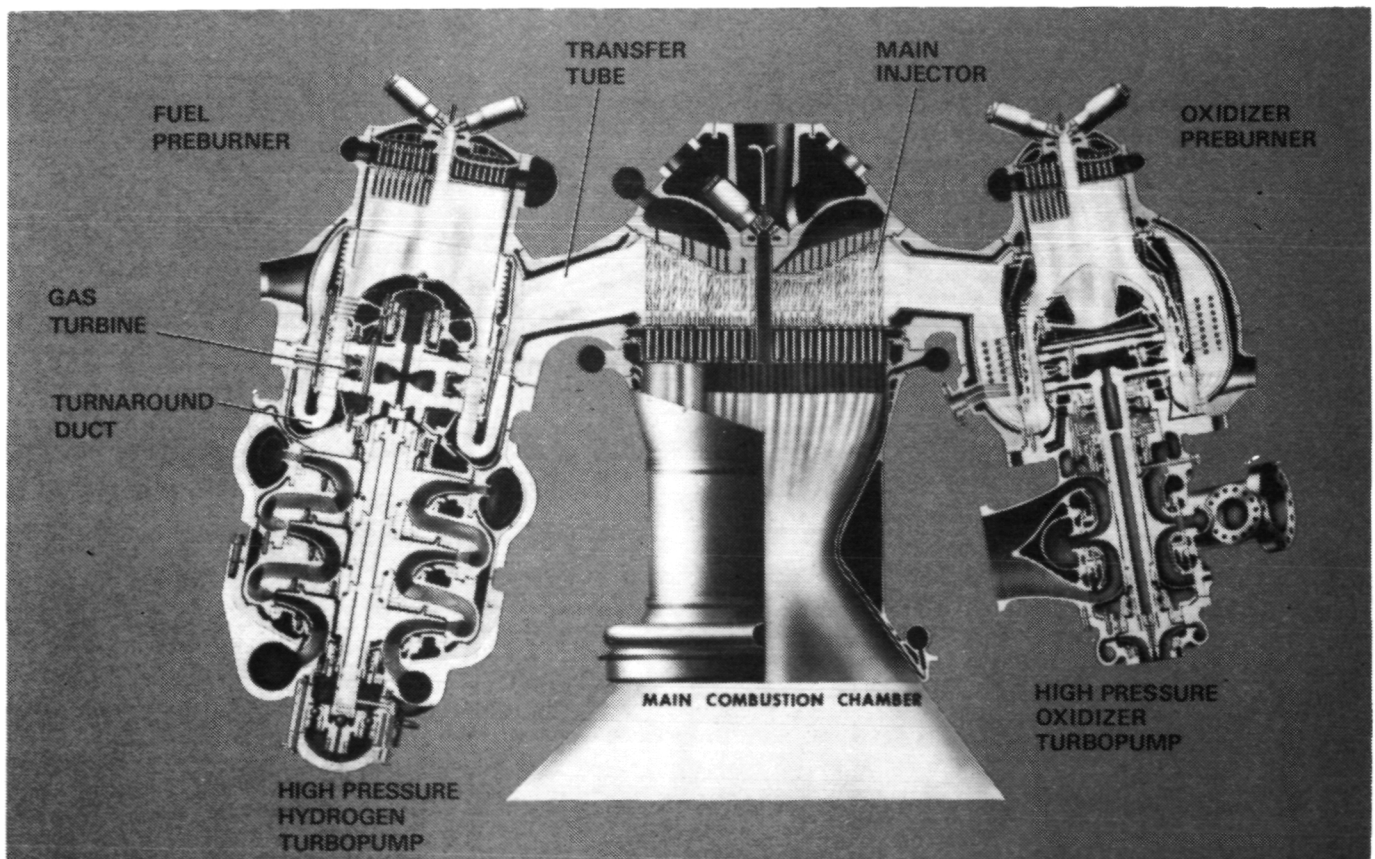


Figure 2.- Schematic of the SSME power head-component arrangement.

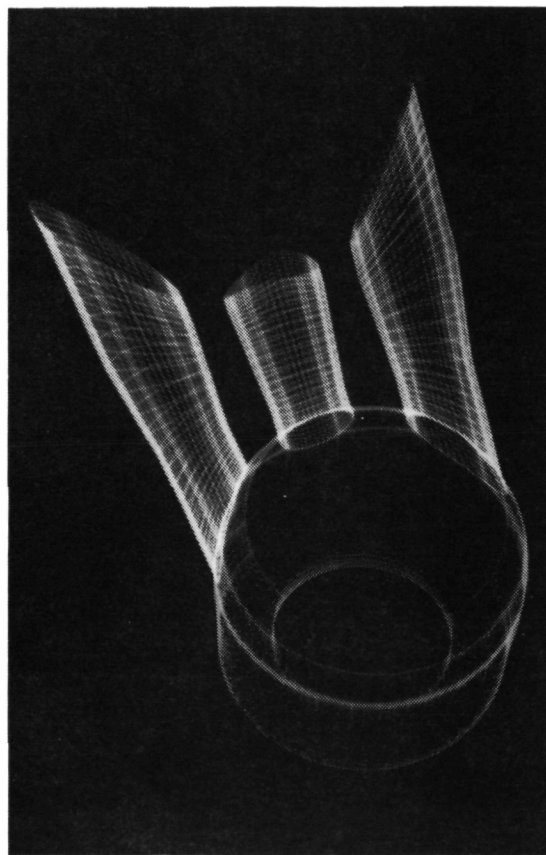
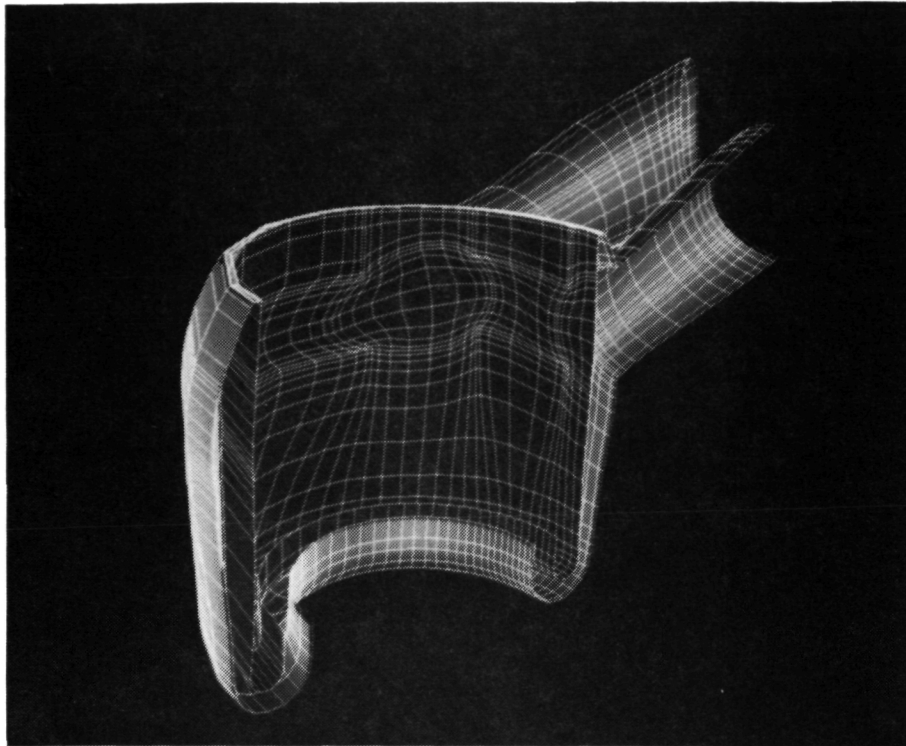


Figure 3.- Computer-generated images of the hot-gas manifold geometry showing the computational mesh on the surfaces. (a) Cross-sectional view; (b) perspective view.

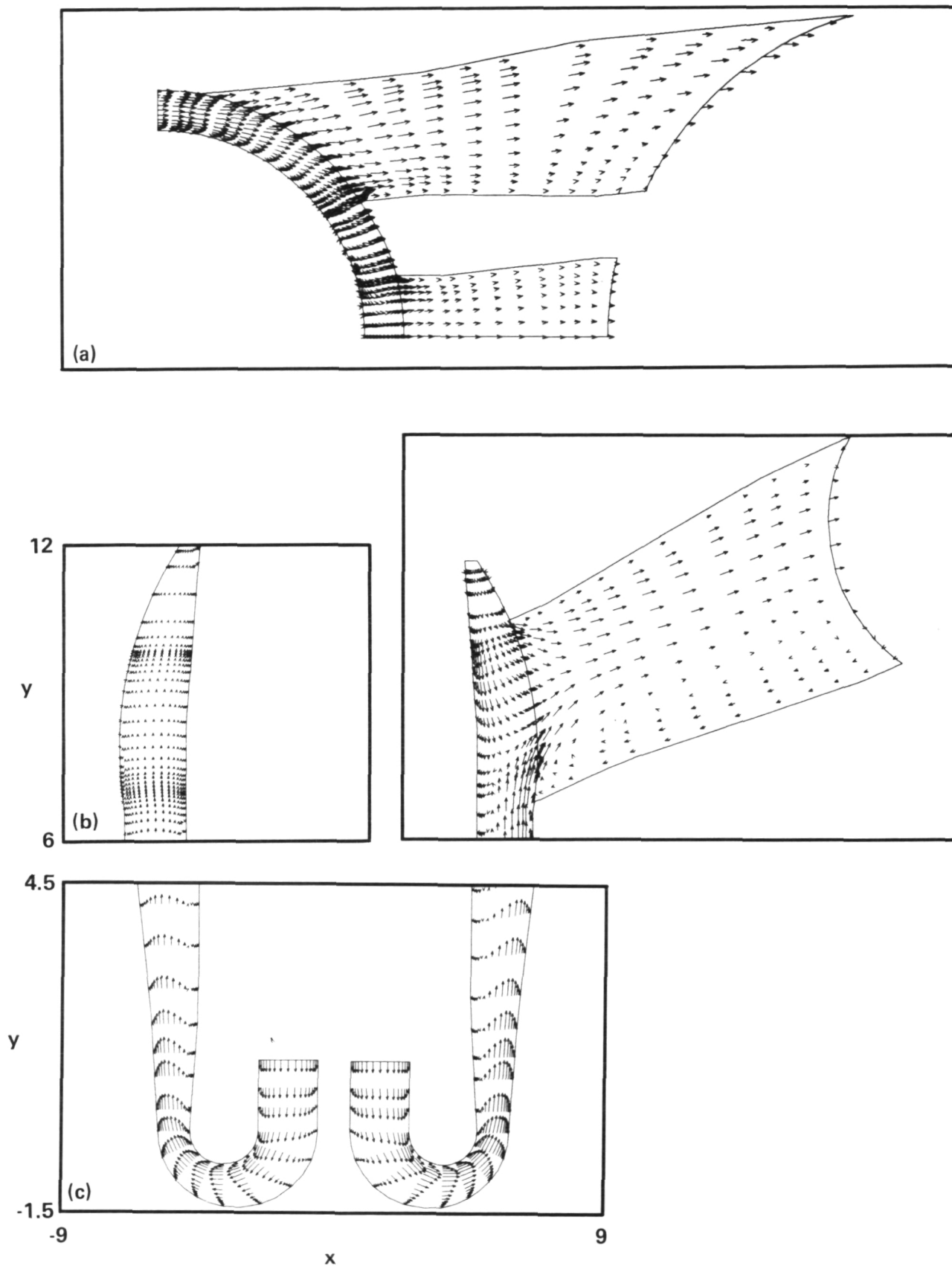


Figure 4.- Computed velocity vectors for the flow through the turnaround duct, fuel bowl, and center transfer tube. (a) Top view; (b) fuel bowl and center transfer tube; (c) annular turnaround duct.

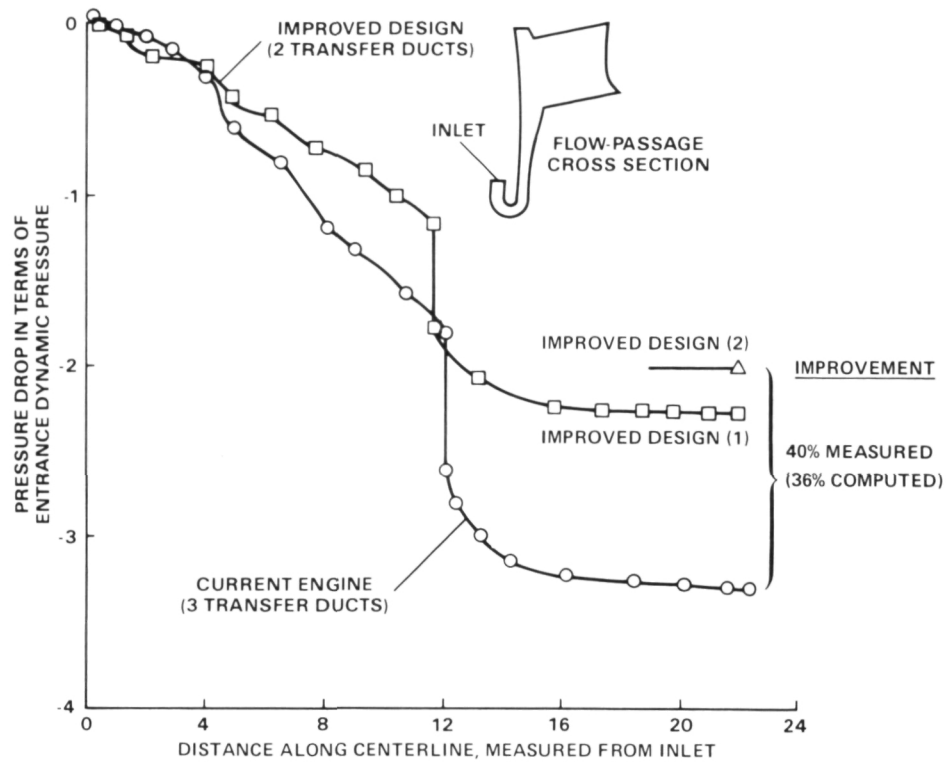


Figure 5.- Measured pressure losses in original and computationally improved hot-gas manifold.

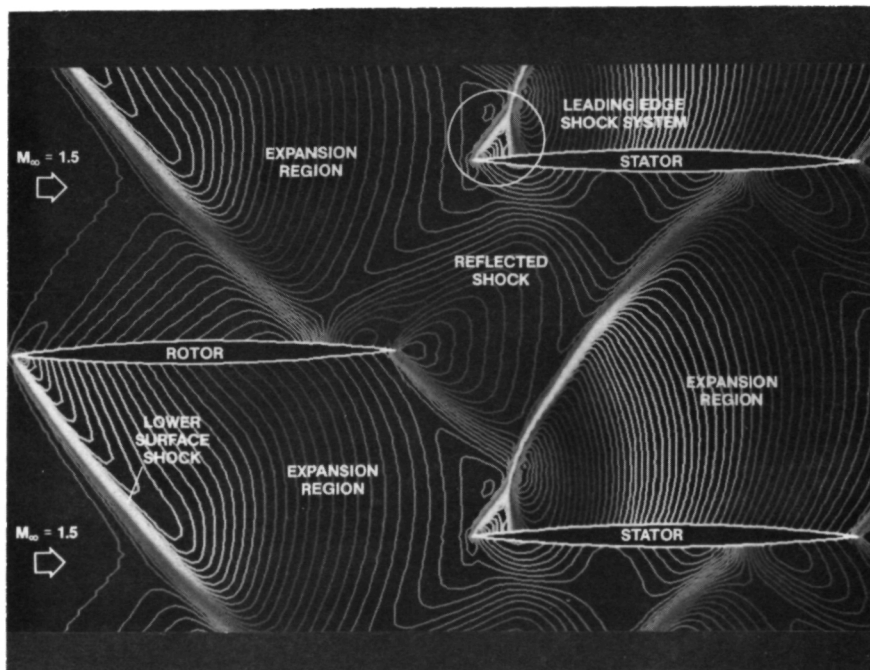


Figure 6.- Instantaneous pressure contours about rotor-stator blades in an axial-flow turbine.

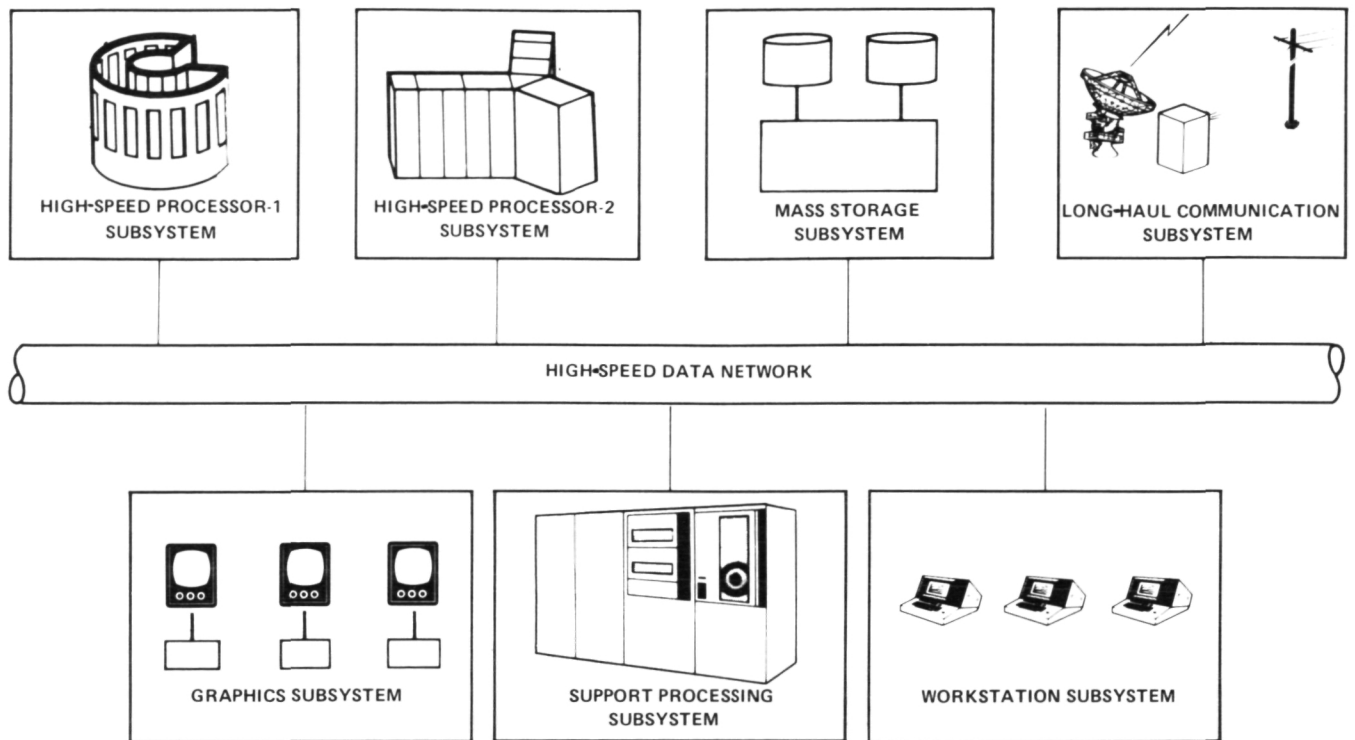


Figure 7.- Schematic of fully developed NPSN.

1. Report No. NASA TM-86769		2. Government Accession No.		3. Recipient's Catalog No.	
4. Title and Subtitle COMPUTATIONAL FLUID DYNAMICS AT NASA AMES AND THE NUMERICAL AERODYNAMIC SIMULATION PROGRAM				5. Report Date September 1985	
				6. Performing Organization Code	
7. Author(s) Victor L. Peterson				8. Performing Organization Report No. 85321	
9. Performing Organization Name and Address Ames Research Center Moffett Field, CA 94035				10. Work Unit No.	
				11. Contract or Grant No.	
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546				13. Type of Report and Period Covered Technical Memorandum	
				14. Sponsoring Agency Code 505-31-01	
15. Supplementary Notes Point of contact: Victor Peterson, Ames Research Center, MS 200-4, Moffett Field, CA 94035, (415) 694-5065 or FTS 464-5065					
16. Abstract Computers are playing an increasingly important role in the field of aerodynamics such that they now serve as a major complement to wind tunnels in aerospace research and development. Factors pacing advances in computational aerodynamics are identified, including the amount of computational power required to take the next major step in the discipline. The four main areas of computational aerodynamics research at NASA Ames Research Center which are directed toward extending the state of the art are identified and discussed. Example results obtained from approximate forms of the governing equations are presented and discussed, both in the context of levels of computer power required and the degree to which they either further the frontiers of research or apply to programs of practical importance. Finally, the Numerical Aerodynamic Simulation Program--with its 1988 target of achieving a sustained computational rate of 1 billion floating-point operations per second--is discussed in terms of its goals, status, and its projected effect on the future of computational aerodynamics.					
17. Key Words (Suggested by Author(s)) Computational aerodynamics Fluid aerodynamics Supercomputers Numerical aerodynamics simulation				18. Distribution Statement Unlimited Subject category - 34	
19. Security Classif. (of this report) Unclassified		20. Security Classif. (of this page) Unclassified		21. No. of Pages 23	
				22. Price* A02	