NASA Technical Memorandum 100289

Internal Fluid Mechanics Research on Supercomputers for Aerospace Propulsion Systems

Brent A. Miller, Bernhard H. Anderson,
and John R. Szuch
Lewis Research Center
Cleveland, Ohio

NASA

INTERNAL FLUID MECHANICS RESEARCH ON SUPERCOMPUTERS FOR AEROSPACE PROPULSION SYSTEMS

Brent A. Miller, Bernhard H. Anderson, and John R. Szuch

National Aeronautics and Space Administration, Lewis Research Center,
Cleveland, Ohio 44135

Abstract. The Internal Fluid Mechanics Division of the NASA Lewis Research Center is combining the key elements of computational fluid dynamics, aerothermodynamic experiments, and advanced computational technology to bring internal computational fluid mechanics (ICFM) to a state of practical application for aerospace propulsion systems.

The strategies used to achieve this goal are to (1) pursue an understanding of flow physics, surface heat transfer, and combustion via analysis and fundamental experiments, (2) incorporate improved understanding of these phenomena into verified three-dimensional CFD codes, and (3) utilize state-of-the-art computational technology to enhance the experimental and CFD research.

This paper presents an overview of the ICFM program in high-speed propulsion, including work in inlets, turbomachinery, and chemical reacting flows. Ongoing efforts to integrate new computer technologies, such as parallel computing and artificial intelligence, into the high-speed aeropropulsion research are described.

INTRODUCTION

Computational Fluid Dynamics (CFD) is becoming an increasingly powerful tool for the aerodynamic design of aerospace systems. This is particularly important in propulsion due to the lack of alternatives for gaining insight into the complicated, highly coupled fluid dynamics within modern propulsion systems.

Improvements in numerical algorithms, geometric modeling, grid generation, and parameter modeling, as well as dramatic improvements in supercomputer processing speed and memory, are now allowing CFD researchers to address more complex configurations and geometries, broader flight regimes, and new applications. These advancements are also permitting models and codes to be combined to solve larger, more complex systems.

For CFD to be used with confidence in propulsion system design, it is necessary to verify and validate existing and forthcoming CFD application codes with the best available experimental data. Because of the complexity of the aerodynamic flows that will be routinely computed, the experimental data base must expand to include not only surface-measurable quantities, but also detailed measurements of fluid and thermal parameters throughout the flow regime of interest. Also, the errors inherent in experimental testing must be identified, understood, and minimized in order to produce the high-quality, benchmark data sets that are required for validation of CFD applications codes and for use in developing physical modeling data.

The process of conducting "benchmark" experiments that highlight one or more basic flow mechanisms and tieing the experiments to computer analyses has had a long history at NASA Lewis [1]. The Internal Fluid Mechanics Division of NASA Lewis is combining the key elements of computational fluid dynamics, aerothermodynamic experiments, and advanced computational technology to bring internal computational fluid mechanics (ICFM) to a state of practical application for aerospace propulsion systems.

The strategies used to achieve this goal are to (1) pursue an understanding of flow physics, surface heat transfer, and combustion via analysis and fundamental experiments, (2) incorporate improved understanding of these phenomena into verified three-dimensional CFD codes, and (3) utilize state-of-the-art computational technology to enhance the experimental and CFD research.

By focusing on specific portions of a propulsion system, it is often possible to isolate and study the dominant phenomena in order to gain the understanding needed to develop an accurate, predictive capability. As shown in Fig. 1, the NASA Lewis ICFM program is organized with Inlets, Turbomachinery, and Chemical Reacting Flows as the main research thrusts.

The development of an accurate, predictive capability in propulsion CFD can only be accomplished if the numerical code development work and the experimental work are closly coupled. As shown in Fig. 2, each discipline is dependent on the other for information and guidance. The insights gained through the experiments form the basis for new mathematical models and codes. The resultant codes must be tested by comparison of analytical and experimental data, often leading to new test requirements.

1

This paper presents an overview of the ICFM program in high-speed propulsion, which includes sustained supersonic cruise, hypersonic to orbit, and advanced high thrust to weight core technology. CFD research activities in inlets, turbomachinery, and chemical reacting flows are described along with ongoing efforts to integrate new computer technologies, such as parallel computing and artificial intelligence, into the high-speed aeropropulsion research.

ICFM RESEARCH THRUSTS

Inlets

With the availability of supercomputers, analyses are now being used in conjunction with experiments to understand the complex flow phenomena present in aircraft inlets. Inlet flows are highly three-dimensional, often with regions of mixed supersonic and subsonic flow and containing steep gradients in the flow variables near shock waves and boundary layers. Detailed experiments and fast, large capacity computers are required to resolve and accurately compute these flowfields.

An example of integrated experiment and analysis is the development of an inlet for the Mach 5.0 cruise aircraft shown in Fig. 3. This vehicle was proposed by Lockheed to the Air Force as a successor to the SR-71. The aircraft has wing-mounted ramjets to provide the required propulsion at Mach 5.0. The proposed design has been studied for several years using both computer simulations and wind tunnel tests of sub-scale inlet models. A picture of one such model is shown in Fig. 4, mounted in the NASA Lewis 1x1 supersonic tunnel. The inlet has rectangular cross section, four external compression ramps located on the bottom, and contoured internal supersonic compression from the cowl at the top. Long swept sideplates run from the leading edge of the inlet to the cowl to prevent compressed flow from spilling over the sides of the inlet.

The sub-scale model was recently tested to verify the computer results shown in Fig. 5. The calculations were done on a Cray X-MP supercomputer. The calculations used a previously verified supersonic PNS analysis [2] with nearly ten million grid points and consumed 5 hours of CPU time. Figure 5 shows planes of color-coded pitot pressure contours in the supersonic portion of the inlet. The near sideplate and cowl have been made transparent to allow easier veiwing of the computed results. The concentrated lines parallel to the ramps of the inlet indicate shock locations while the concentration of blue lines on the ramp and sidewall surfaces indicate boundary layer distributions. The computed side-wall boundary layer is thick and highly distorted by the sweeping action of the compression shocks, causing the flow to separate in the corner formed by the cowl and sideplate just inside the cowl. A detailed picture of this last plane is shown in Fig. 6. On the left side of the figure is color-coded Mach number and on the right is secondary velocity vectors. The high distortion produced by the vortex-like structure in the corner would cause serious problems for the ramjet burners, while the corner separation might trigger an inlet unstart for this design. The sub-scale model of Fig. 4 has encoun-

tered some of the problems predicted by the computer analysis during testing.

Because the computer results were obtained early in the design process, it was possible to redesign the inlet to incorporate boundary layer control devices in the vicinity of these predicted problems. Tests of the redesigned full-scale inlet model will be conducted in the NASA Lewis 10x10 supersonic wind tunnel.

To fully understand and visualize the massive amounts of data produced by supercomputer analyses, one needs to employ some type of computer graphics workstation. Figure 7 shows a researcher working at a Silicon Graphics Iris workstation that is connected to the Numerical Aerodynamic Simulator (NAS) computing network. In the figure, the researcher is shown comparing computed surface oil flows in a glancing shock boundary layer interaction with results from an experiment at the identical flow condition. The use of three-dimensional visualization techniques is resulting in a better understanding of experimental results and the underlying physics. This is enabling the development of computer codes that can accurately predict detailed flow features thus increasing confidence in the use of the codes for propulsion system design.

Turbomachinery

Current research in turbomachinery CFD at NASA Lewis is focused on providing verified computer codes for the agressive design of advanced components for the next century's propulsion systems. Turbomachinery geometries will include endwall contouring and leaned, bowed, and swept blading. The flow physics within these components will be extremely complex due to the presence of unsteady rotor-stator interactions and large secondary flows.

A range of computational tools is envisioned for advanced turbomachinery design and analysis. Figure 8 shows the relative levels of complexity of these tools, ranging from the unaveraged Navier-Stokes equations down to the quasi-one-dimensional equations. Since the flow field within advanced turbomachinery is highly three-dimensional and fundamentally unsteady, three-dimensional, unsteady, viscous codes will be needed to predict the flow field.

The major problem to be addressed in building and applying these codes is determining the appropriate level and type of temporal and spatial averaging. The unaveraged, three-dimensional Navier-Stokes equations do not involve any averaging and are the most complete and complex representation of the unsteady flow field. The Reynolds-averaged, Navier-Stokes equations involve time-averaging of the turbulent fluctuating components and require empirical closure models for turbulence. These equations can be further time-averaged to produce the steady, time-averaged equations. Additional space-averaging can then be applied to produce average-passage equations, the axisymmetric equations, and, finally, the quasi-one-dimensional equations. While much less information is available from the quasi-one-dimensional equations, they can produce useful

2

information without a large expenditure of computational resources if accurate empirical models are carefully employed. The development of the empirical models and the verification of the code are based upon a range of experimental measurements.

Figure 9 shows the relative levels of complexity of experimental measurements as one attempts to obtain more and more information in the time domain. Time-resolved measurements using hot-wire anemometry offer the most information about the unsteady flow field. However, the challenges listed in Fig. 10 usually make nonintrusive, laser anemometry the method of choice for obtaining high resolution data in turbomachines. It provides ensemble averaging of random statistics while retaining and identifying the deterministic time dependency. In order to overcome the long measurement times required by laser anemometry and to capitalize on the detailed nature of the data captured, computer control of data acquisition and real-time data reduction and display are required. NASA Lewis is recognized as a world leader in the development and application of computer-controlled laser anemometry for turbomachinery experimental research. Two examples of this research are discussed below. The first involves the calculation of rotor-stator interaction effects in a turbine stage. The second involves the measurement of kinetic energy distribution within a compressor stator.

Turbine Rotor-Stator Interaction Calculation. A quasi-three-dimensional, viscous code for isolated blade rows was modified to predict rotor-stator interaction in a turbine stage [3]. The geometry was based upon the first stage of the Space Shuttle Main Engine (SSME) high-pressure fuel turbopump. A converged, periodic solution was obtained after the stator had seen 10 pitch rotations of the rotor. This solution required approximately 2.5 hr on a Cray X-MP supercomputer for a stator grid of 115 X 31 points and a rotor grid of 197 X 41 points. Mach contours from the solution are shown in Fig. 11 for one instant of time within a blade-passing period. The average stator inlet Mach number is 0.15. The wake region that develops downstream of the stator passes through the grid interface into the inlet portion of the rotor passage. Disturbances from the passing rotor blades also affect the flow within the upstream stator passage. This unsteady method allows important physical effects to be captured that would not appear in a steady code.

Compressor Turbulent Kinetic Energy Measurements. Detailed measurements of the unsteady flow field within a compressor stator, operating downstream of a transonic fan rotor, were obtained using laser anemometry [4]. Figure 12 shows the ensemble-averaged, unresolved unsteadiness in the stator flow field for one relative rotor-stator position. The unresolved unsteadiness includes unsteadiness due to both turbulence and vortex shedding. Areas of high unresolved unsteadiness contain fluid in the rotor blade wake. As the rotor blades rotate past the stator blades, the rotor wakes are convected through the stator blade passages and, subsequently, chopped by the stator blades.

Data obtained at other instances in time during the blade-passing cycle have been used to produce a motion picture which illustrates the ensemble-averaged wake dynamics and its effect on the stator flow field. This information is very valuable for developing closure models and verifying codes.

Chemical Reacting Flows

Future aerospace propulsion concepts involve the combustion of liquid or gaseous fuels in a highly turbulent, internal air stream. Accurate computer codes to predict such chemical reacting flows will be a critical element in the design of these concepts. Supercomputers will be needed due to the large memory and long run-time requirements.

Current research in chemical reacting flows at NASA Lewis is divided into three major areas as shown in Fig. 13. The first area, Fluid Mechanics, involves the study of the basic fluid flow phenomena associated with combustion without the added complexity of heat release. Research in this area includes the development of computer codes for the multiphase processes of fuel sprays. The second area, Combustion Chemistry, concentrates on the combustion of fuel and oxidizer without including the fluid mechanics. Research in this area includes the development of accurate combustion models for use in fluid mechanics codes. The third area, Turbulence-Combustion Interaction, deals with the interactive effects of fluid mechanics and combustion chemistry. Research in this area includes the development of direct numerical simulation techniques for the entire combustion process.

Two examples of current research in chemical reacting flows are discussed below. The first involves experimental measurements in a planar reacting shear layer. The second involves results from a time-accurate, two-dimensional shear layer code.

Planar Reacting Shear Layer Experiment. The objectives of this experiment are to investigate the coupling between fluid mechanics and combustion in a realistic environment and to provide benchmark data for verification of computer codes. The test facility is shown in Fig. 14. The hydrogen-nitrogen stream mixes with the preheated air stream downstream of a plane splitter plate. Combustion occurs where the fuel and air have properly mixed. Pressure oscillations will exist due to the dynamic features of the flow; the coupling between these oscillations and the combusting shear layer will be examined. The unique features of this experiment include: continuous flow capability, high subsonic velocities for both the hydrogen-nitrogen and air streams, uncontaminated preheating of the air stream, and realistic high heat release.

Two-Dimensional Shear Layer Code. A time-accurate, two-dimensional, incompressible, finite-difference code has been used to predict the unsteady development of a forced shear layer [5]. Figure 15 shows the vorticity structure for a Reynolds number of 96 000, based on the forcing wavelength and the

mean convective velocity. The positive and negative vorticity contours originate at the boundary layers specified at the inlet of the computational domain. Forcing is applied at a long wavelength and smaller scale vortices spontaneously develop due to the natural instability of the shear layer. These small scale vortices cluster on the scale of the longer forced wavelength. Small pockets of positive vorticity persist as remnants of the low-speed boundary layer. The collective interaction of the small-scale vortices, as they merge into larger scale structures, controls the dynamics of the developing shear layer. These calculations were performed on the NAS Cray-2 supercomputer.

ADVANCED COMPUTATIONAL TECHNOLOGIES

Computer Capabilities and Requirements

While the speed and memory capacity of computers have increased dramatically over the past 40 years, computer technology still remains a limiting factor in the quest to use high-fidelity computer simulations as tools in the propulsion system design process.

Given the current state of simulation models and computing capability, propulsion systems are being designed, for the most part, using simplified, highly empirical models. If one wanted to use a time-accurate, three-dimensional Navier-Stokes code to design a hypersonic inlet, it is estimated that it would take on the order of 100 hr of CPU time on a CRAY 2 supercomputer to obtain a single flow field solution. If a large number of repetitive runs are required (e.g. design optimization), the use of such a code would be impractical. One would need a speed increase of 2 to 3 orders of magnitude (see Fig. 16) to bring the CPU time down to a more reasonable 1 to 2 hr. A similar shortfall exists in the simulation of complex three-dimensional flows in combustors, compressors, turbines, and nozzles.

When faced with the need for revolutionary, rather than evolutionary designs (as in the case of the National Aerospace Plane), the empirical approach to propulsion system design would appear to be inadequate. Advances in computational technology and the ability to apply that technology are needed if we hope to satisfy the need for and fulfill the promise of computer-aided propulsion system design.

Impact of New Technologies

Supercomputers are just one of many computer technologies that can make ICFM practical for propulsion system design. As shown in Fig. 17, the code development process can benefit from advances being made in parallel processing, artificial intelligence/expert systems, graphics, data communications, and data base management. If used properly, these technologies can lead to a new way of conducting ICFM research with analyses and experiments carried out in a much more integrated and cooperative fashion. The resulting databases, codes, and insights can form the basis for a more confident, more aggressive design methodology.

The following paragraphs describe efforts at NASA Lewis to develop and apply advanced computational technologies to ICFM.

Local-Area Networks. In 1985, work began on the design of a local-area network (ERBNET) that would allow ICFM researchers to access and communicate between various terminals, computers, and workstations located in the Engine Research Building (ERB) complex. The network would also provide connectivity and high-speed data communications between the local computers, the NASA Lewis central mainframe computers, and remote computers such as the NAS.

In designing the ERBNET system, special attention was given to system adaptability, flexibility, and growth potential. The decision was made to use proven and available technology (Ethernet communications and TCP/IP networking protocols) to reduce cost, allow rapid implementation, and permit modular upgrades.

Figure 18 shows a block diagram representation of the ERBNET configuration. A baseband Ethernet cable was installed in the ERB to provide high-speed (10 Mbits/sec) connections between the local computers. A utility band on the NASA Lewis-wide broadband cable (LINK) was used to interconnect ERBNET with computers in the central Research Analysis Center (RAC) and to provide user access to the NAS facilities through the NASA Program Support Communications Network (PSCN).

Plans call for ERBNET to be expanded to provide additional capabilities to a greater number of users. For example, ERBNET will be tied into off-site networks to allow sharing of resources and information with university and industrial researchers. Additional equipment is continually being added to the network, including additional high-performance graphics workstations. Work is underway to develop graphics and scientific database software that will facilitate the interchange and display of data across the breadth of ERBNET devices.

Artificial Intelligence. IFMD is currently investigating the use of Artificial Intelligence (AI) concepts as aids to researchers developing and using flow-solver codes. Such codes are typically written by users with specific problems in mind and are consequently difficult for others to use and extend to other problems. The effort at NASA Lewis attempts to capture the expertise gained by code developers in an "intelligent interface" that will allow others to make more effective use of the codes.

The interface is initially being developed to support the PROTEUS code (a general-purpose three-dimensional Navier-Stokes flow solver being developed at NASA Lewis using software engineering principles). The interface has been named PROTAIS to denote the introduction of AI techniques to the PROTEUS code. As shown in Fig. 19, the PROTAIS system provides help and advise for both expert and novice users of PROTEUS. It features a constantly growing knowledge base that reflects the experience and knowledge gained as the PROTEUS code is developed and run under a variety of conditions.

4

Work, to date, has concentrated on developing the expert user facilities within PROTAIS. This constitutes the basic foundation of the interface, automating many of the tedious and time-consuming aspects of setting up and running the code. The PROTAIS code development is being done on a Symbolics Workstation (see Fig. 20). At this writing, testing of the software design is being carried out in a stand-alone mode (simulating the interface to PROTEUS). In 1988, the ERBNET facilities will be used to connect the interface, installed on workstations, to the PROTEUS code that is running on an IBM 3033 mainframe. The interface will then undergo testing and evaluation by the PROTEUS code developers.

Parallel Processing. Advancements in solid-state electronics and circuit design technology produced nearly linear increases in computer speeds from 1950 to 1970. Since that time, computer manufacturers have had to look toward new computer architectures (e.g., vector and parallel processors) to gain additional computing speed. The current generation of supercomputers makes use of multiple, bus-connected vector processors with shared memory to achieve 100 Mflop-level speeds. Recently, Mulac, et al. [6] have been successful in using four processors on the CRAY X-MP/416 to calculate the flow field in a multistage turbine.

Aerospace researchers can expect this trend in supercomputer technology to continue with the next generation of systems employing 8 to 16 high-speed processors to move closer to gigaflop performance However, the high cost of supercomputers and the need to share the available CPU time among many users (each wanting to run large, time-and-memory-consuming codes), often make supercomputers unavailable and/or impractical for many time-critical applications, such as real-time simulation and on-line data processing.

Recently, parallel architectures and state-of-the-art microprocessors have been combined to form a new class of machine – the minisupercomputer. Mini-supercomputers offer an attractive alternative to mini-computers with some systems delivering near-supercomputer speeds (10 to 20 Mflop) and costing less than $1M. A number of studies [7-8] are currently underway to investigate the use of mini-supercomputers for solving ICFM algorithms.

At NASA Lewis, parallel computing will be used to mold a more synergistic ICFM research program. By providing a parallel-processing "compute engine" for both analysts and experimentalists (see Fig. 21), we hope to facilitate and accelerate progress in developing more powerful, validated codes. Experimentalists will be able to use parallel processing to speed up the collection, manipulation, and viewing of data. The CFD codes, themselves, should become commonly-used tools in planning, guiding, and interpreting results from experiments.

NASA Lewis researchers are now working to determine the proper mix of algorithms and architectures for selected ICFM applications. To support this research, a reconfigurable, parallel processing workstation, called the "Hypercluster", is being

constructed [9]. The Hypercluster will allow researchers to conveniently implement both shared memory and distributed memory architectures and to interactively study new algorithmic approaches. An earlier version of the workstation is shown in Fig. 22. Assembly and testing of a 16-processor Hypercluster is expected to be completed in late 1988. Applications to be studied include ICFM flow solvers (analysis) and parallel processing of laser anemometry data (experiments).

CONCLUDING REMARKS

Improvements in numerical algorithms, geometric modeling, grid generation, and parameter modeling, as well as dramatic improvements in supercomputer processing speed and memory, are now making CFD a powerful tool for the aerodynamic design of propulsion systems. Validation of existing and forthcoming models and codes is needed to gain confidence in the use of the CFD codes as design tools. The code validation can best be accomplished by a close coupling of "benchmark" experiments, highlighting one or more basic flow mechanisms, and CFD code development. Proper use of rapidly advancing computational technologies, including supercomputers, parallel processors, graphics, artificial intelligence, and networks, can greatly enhance our capabilities to conduct this research. The Internal Fluid Mechanics Division of NASA Lewis is combining the key elements of computational fluid dynamics, aerothermodynamic experiments, and advanced computational technology to bring internal computational fluid mechanics (ICFM) to a state of practical application for aerospace propulsion systems.

REFERENCES

[1] Anderson, B.H.: Three-Dimensional Viscous Design Methodology for Advanced Technology Aircraft Supersonic Inlet Systems, AIAA Paper 84-0194, January 1984.

[2] Benson, T.J.: Three-Dimensional Viscous Calculation of Flow in a Mach 5.0 Hypersonic Inlet, AIAA Paper 86-1461, 1986.

[3] Jorgenson, P.C.E., and Chima, R.V.: An Explicit Runge-Kutta Method for Unsteady Rotor/Stator Interaction, AIAA Paper 88-0049, January 1988.

[4] Hathaway, M.D.: Unsteady Flows in a Single-Stage Transonic Axial-Flow Fan Stator Row, Ph. D Thesis, Iowa State Univ. NASA TM-88929, 1987.

[5] Claus, R.W., Huang, P.G., and MacInnes, J.: Time-Accurate Simulations of a Shear Layer Forced at a Single Frequency, AIAA Paper 88-0061, January 1988.

[6] Mulac, R.A., et al , "The Utilization of Parallel Processing in Solving the Inviscid Form of the Average-Passage Equation System for Multistage Turbomachinery," In: Computational Fluid Dynamics Conference, 8th, Honolulu, Hawaii, June 9-11, 1987, pp. 70-80.

[7] Modiano, D.: Performance of a Common CFD Loop on Two Parallel Architectures, MIT CFD Laboratory Report CFDL-TR-87-11, November 1987.

[8] Smith, W.A.: Multigrid Solution of Euler Equations, Ph. D Thesis, Cornell University 1987.

[9] Blech, R.A.: The Hypercluster: A Parallel Processing Test Bed Architecture for Computational Mechanics Applications, NASA TM-89823. (Presented at the Summer Computer Simulation Conference, Montreal, Canada, July 27-30, 1987.)
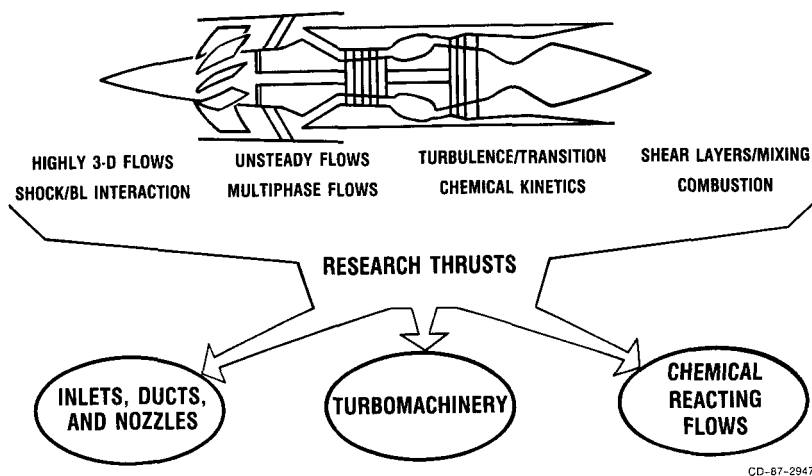
FIGURE 1. - RESEARCH THRUSTS IN INTERNAL COMPUTATIONAL FLUID MECHANICS.
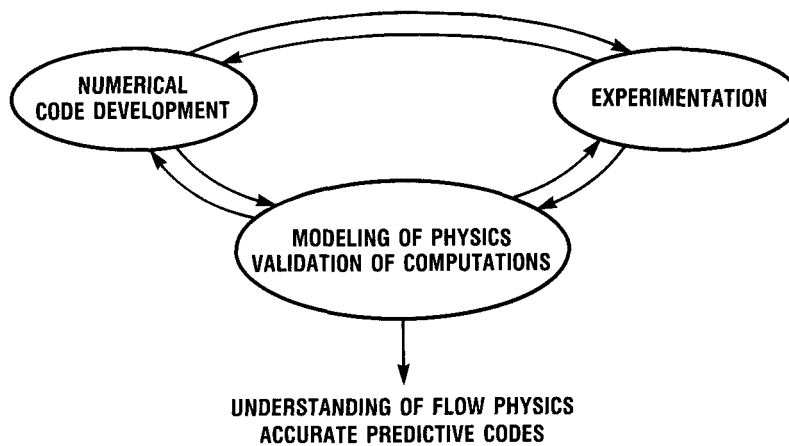
FIGURE 2. - CLOSELY COUPLED EXPERIMENTAL AND COMPUTATIONAL RESEARCH.
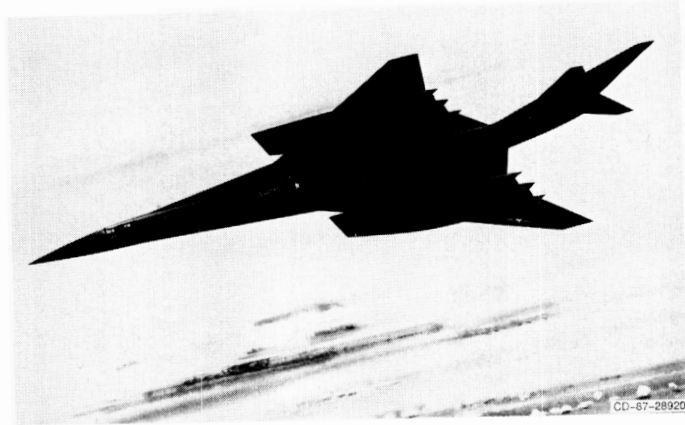
FIGURE 3. – MACH 5 CRUISE AIRCRAFT.

FIGURE 4. – MACH 5 INLET EXPERIMENT.

7
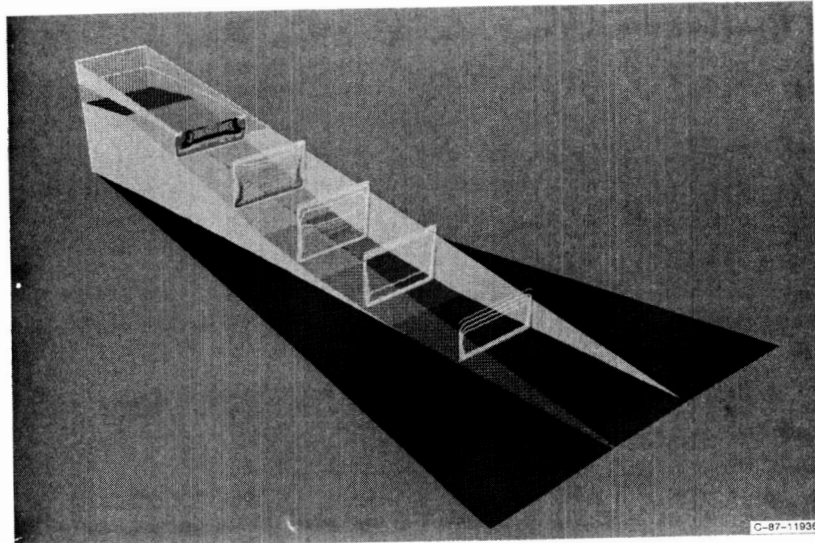
FIGURE 5. – MACH 5 INLET ANALYSIS.
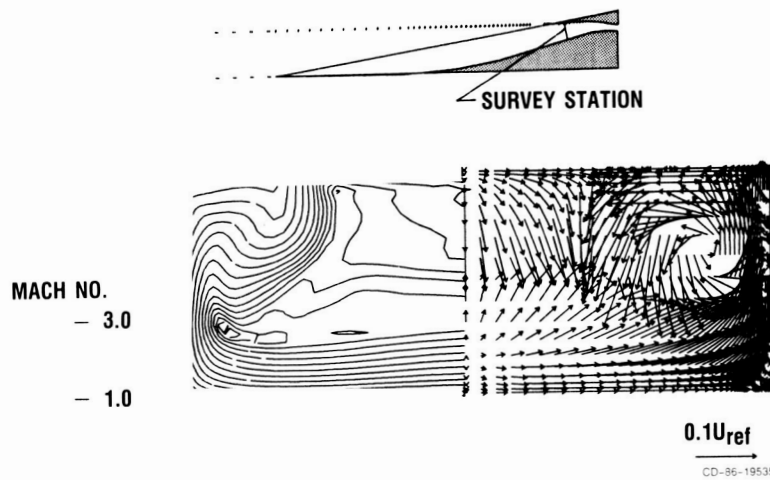
FIGURE 6. – MACH NUMBER AND SECONDARY VELOCITY VECTORS
FOR MACH 5.0 HYPERSONIC INLET.

8

FIGURE 7. – GRAPHICS WORKSTATIONS FOR VISUALIZATION OF 3D FLOW FIELD DATA.
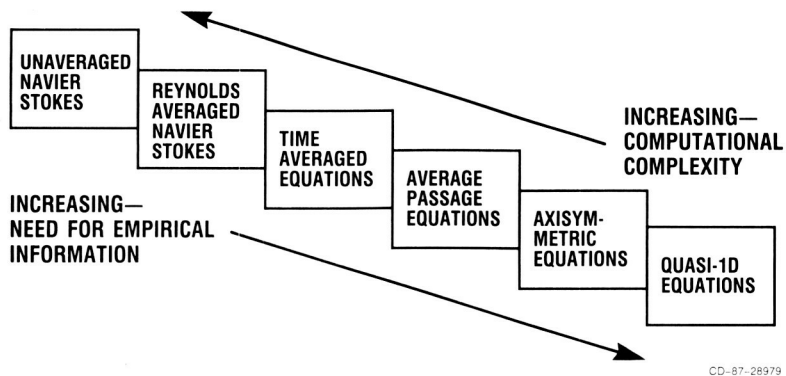
FIGURE 8. – LEVELS OF COMPLEXITY FOR COMPUTATIONAL ANALYSIS.

9

TIME-RESOLVED
MEASUREMENTS
(HOT WIRES)

ENSEMBLE-
AVERAGED
MEASUREMENTS
(LASER)

QUANTITATIVE
TIME-AVERAGED
MEASUREMENTS
(PITOT-STATIC)

QUALITATIVE
FLOW
VISUALIZATION
(SMOKE)

INCREASING—
DETAIL IN THE
TIME DOMAIN

INCREASING—
AVERAGING OF
TEMPORAL BEHAVIOR
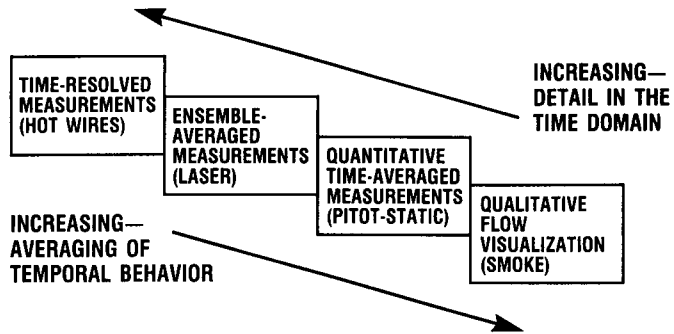
CD-87-28980

FIGURE 9. – LEVELS OF COMPLEXITY FOR EXPERIMENTAL
MEASUREMENTS.

VELOCITY

LASER
ANEMOMETER
OPTICS

CHALLENGES

• ROTATING MACHINERY
• TRANSONIC VELOCITIES
• SMALL BLADE PASSAGES
• 3D FLOWS
• RESTRICTED ACCESS
• HIGH NOISE AND VIBRATION

ON-LINE
DATA
DISPLAY

ROTOR POSITION

COMPUTER
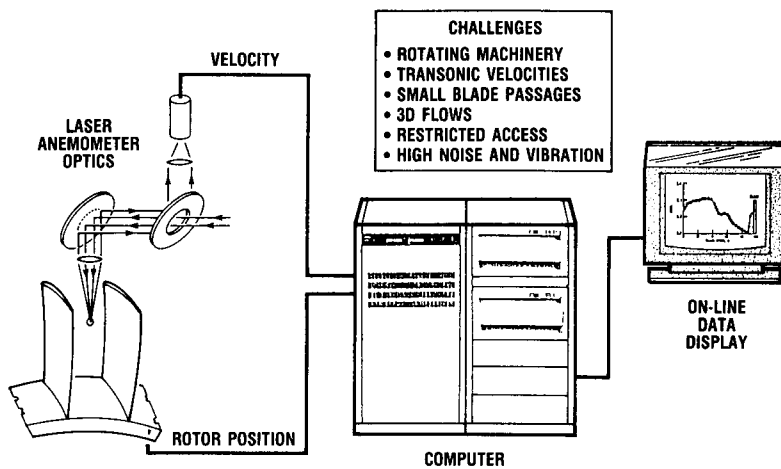
CD-87-28983

FIGURE 10. – TURBOMACHINERY LASER ANEMOMETRY SYSTEMS.

10
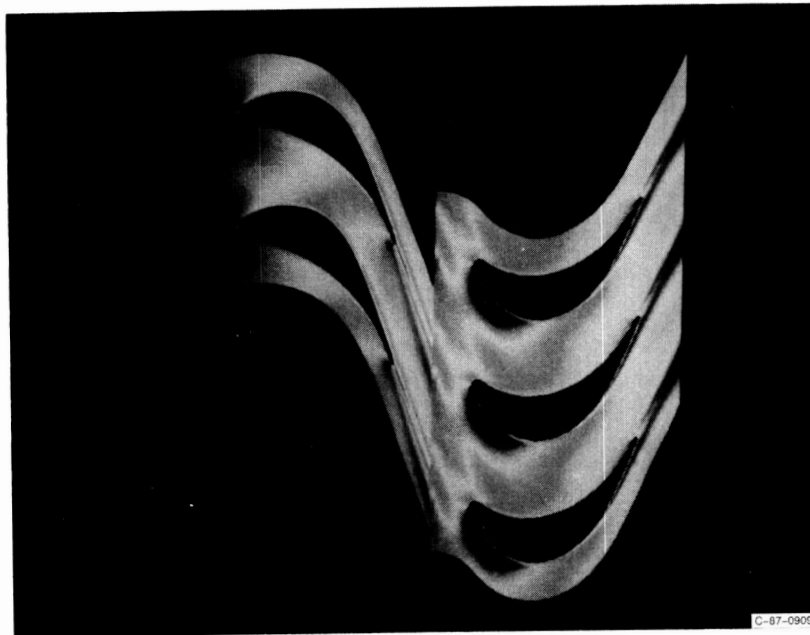
FIGURE 11. – ROTOR-STATOR INTERACTION CALCULATIONS FOR SSME
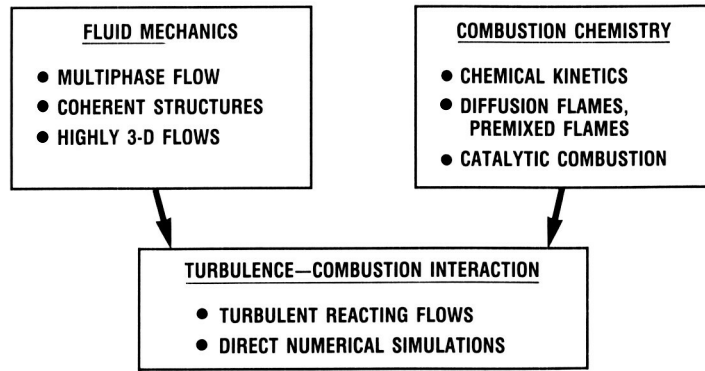FUEL TURBINE—MACH CONTOUR DISTRIBUTION.
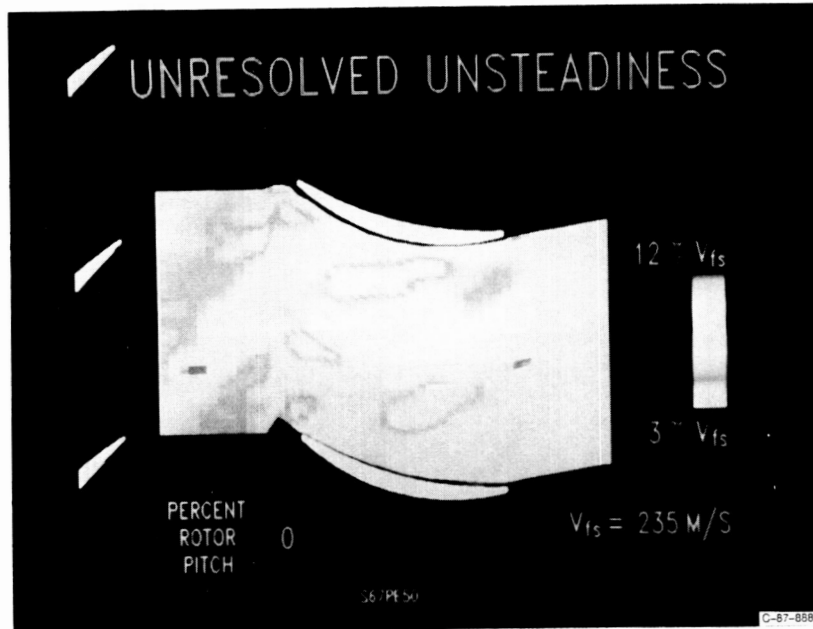
FIGURE 12. – COMPRESSOR "TURBULENT" KINETIC ENERGY.

FLUID MECHANICS	COMBUSTION CHEMISTRY
• MULTIPHASE FLOW	• CHEMICAL KINETICS
• COHERENT STRUCTURES	• DIFFUSION FLAMES, PREMIXED FLAMES
• HIGHLY 3-D FLOWS	• CATALYTIC COMBUSTION

TURBULENCE—COMBUSTION INTERACTION

• TURBULENT REACTING FLOWS
• DIRECT NUMERICAL SIMULATIONS

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

CD-87-28755

FIGURE 13. – ELEMENTS OF CHEMICAL REACTING FLOW RESEARCH.

FEATURES

• CONTINUOUS FLOW
• HIGH SUBSONIC VELOCITY
• NONVITIATED AIR PREHEAT
• HIGH HEAT RELEASE

LASER

HYDROGEN-NITROGEN

AIR

CD-87-28765
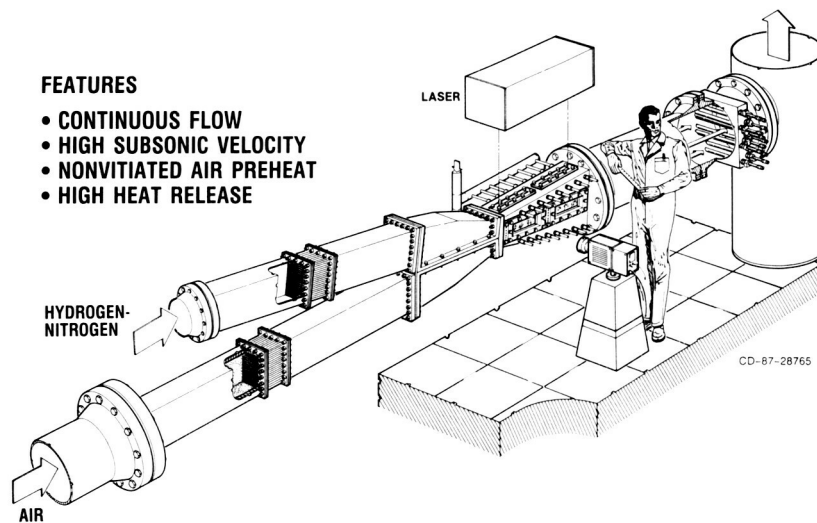
FIGURE 14. – PLANAR REACTING SHEAR LAYER EXPERIMENTS.
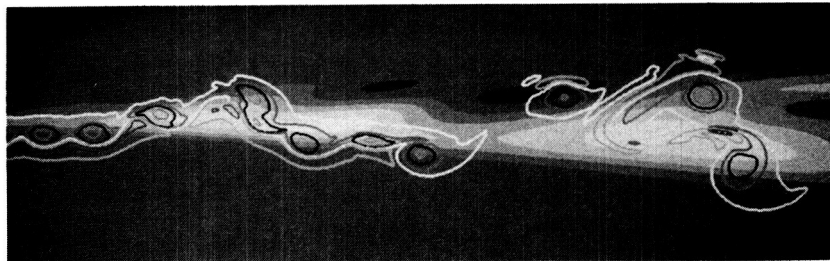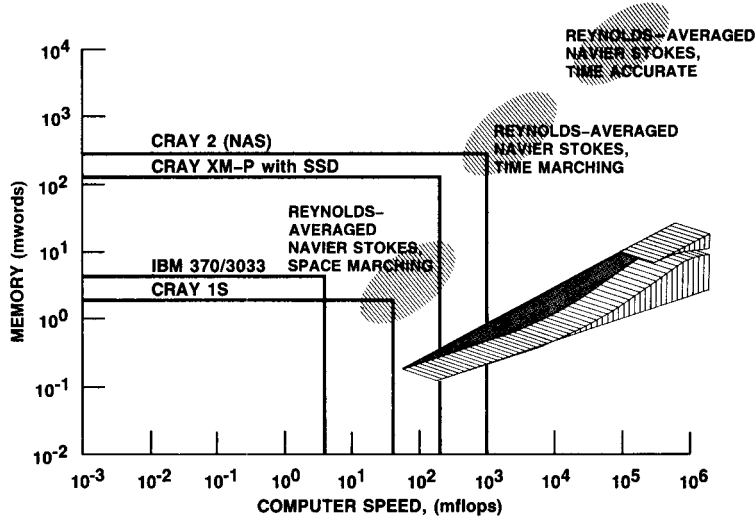

FIGURE 15. – TURBULENT REACTING FLOW CALCULATIONS.

FIGURE 16. – COMPUTER CAPABILITIES AND REQUIREMENTS
FOR 3D HYPERSONIC INLET CALCULATIONS.
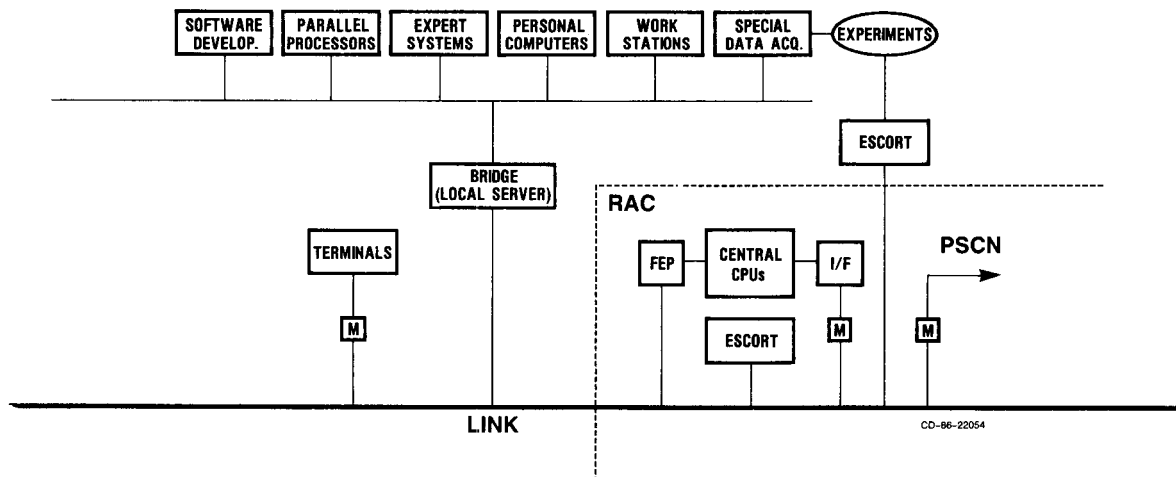
TECHNOLOGIES

- SUPERCOMPUTERS
- PARALLEL PROCESSORS
- EXPERT SYSTEMS
- INTERACTIVE 3-D GRAPHICS
- NETWORKS
- DBMS

IMPROVED CODES

- MORE ACCURATE
- EASIER TO USE
- VALIDATED BY EXPERIMENTS
- PROVIDE NEW PHYSICAL INSIGHTS
- BASIS FOR BETTER, FASTER DESIGNS

FIGURE 17. – FUTURE IMPACT OF COMPUTERS ON INTERNAL
COMPUTATIONAL FLUID MECHANICS.

FIGURE 18. – ERBNET LOCAL AREA COMPUTING NETWORK.

13

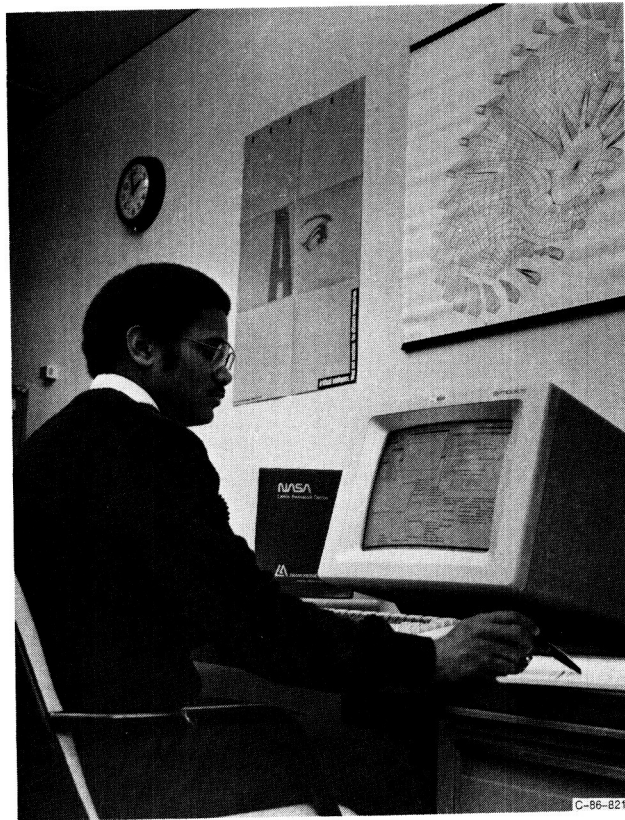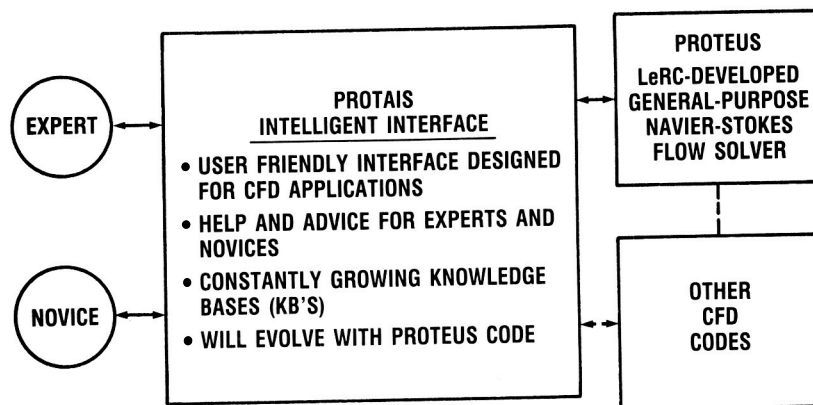FIGURE 19. – ADVANCED INTELLIGENT WORKSTATIONS.

FIGURE 20. – APPLICATION OF ARTIFICIAL INTELLIGENCE TO COMPUTATIONAL FLUID DYNAMICS.

14

CRAY X-MP
NAS

ERBNET

CFD DEVELOPMENT

CLOSURE

PARALLEL COMPUTING

GUIDANCE

EXPERIMENTS
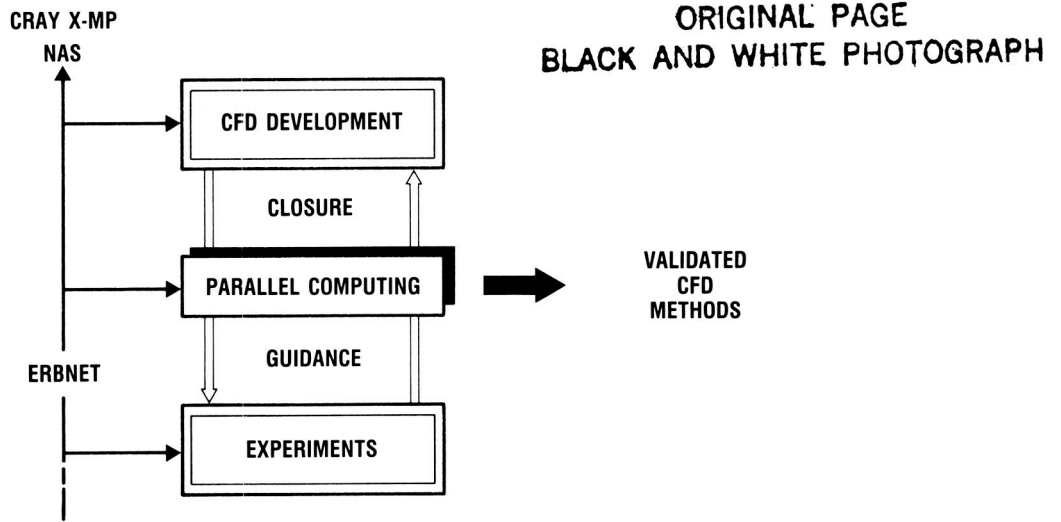
VALIDATED
CFD
METHODS

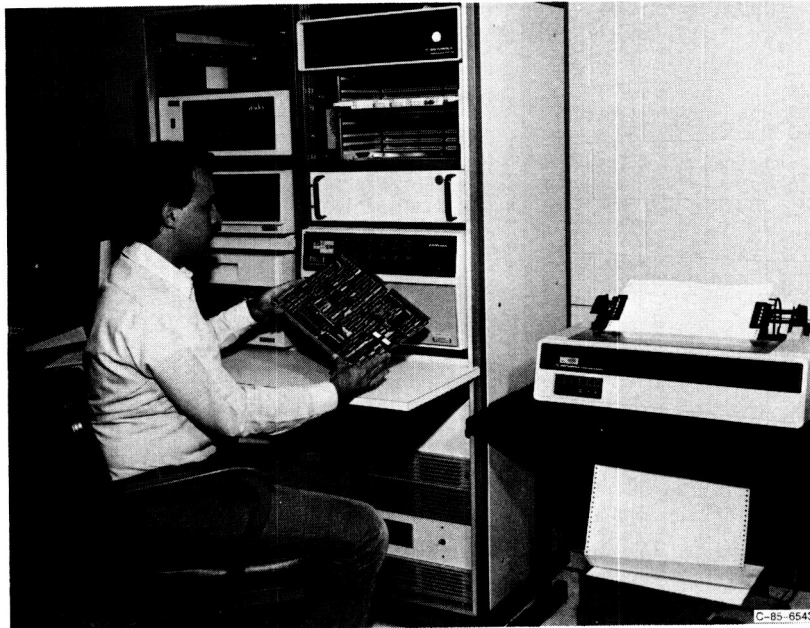FIGURE 21. – PARALLEL COMPUTING FOR VALIDATION OF CFD CODES.

FIGURE 22. – PARALLEL PROCESSING WORKSTATION.

Report Documentation Page

1. Report No. NASA TM-100289	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle Internal Fluid Mechanics Research on Supercomputers for Aerospace Propulsion Systems		5. Report Date January 1988
		6. Performing Organization Code
7. Author(s) Brent A. Miller, Bernhard H. Anderson, and John R. Szuch		8. Performing Organization Report No. E-3937
		10. Work Unit No. 505-62-21
9. Performing Organization Name and Address National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191		11. Contract or Grant No.
		13. Type of Report and Period Covered Technical Memorandum
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, D.C. 20546-0001		14. Sponsoring Agency Code

16. Abstract

The Internal Fluid Mechanics Division of the NASA Lewis Research Center is combining the key elements of computational fluid dynamics, aerothermodynamic experiments, and advanced computational technology to bring internal computational fluid mechanics (ICFM) to a state of practical application for aerospace propulsion systems. The strategies used to achieve this goal are to (1) pursue an understanding of flow physics, surface heat transfer, and combustion via analysis and fundamental experiments, (2) incorporate improved understanding of these phenomena into verified three-dimensional CFD codes, and (3) utilize state-of-the-art computational technology to enhance the experimental and CFD research. This paper presents an overview of the ICFM program in high-speed propulsion, including work in inlets, turbomachinery, and chemical reacting flows. Ongoing efforts to integrate new computer technologies, such as parallel computing and artificial intelligence, into the high-speed aeropropulsion research are described.

17. Key Words (Suggested by Author(s)) Aeropropulsion; Charge flow devices; Simulation; Supercomputers; Flow physics	18. Distribution Statement Unclassified - Unlimited Subject Category 34

19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No of pages 16	22. Price* A02