NASA Technical Memorandum 102132
AIAA-89-2904

CFD in the Context of IHPTET—The Integrated High Performance Turbine Technology Program

(NASA-TM-102132) CFD IN THE CONTEXT OF IHPTET: THE INTEGRATED HIGH PERFORMANCE TURBINE TECHNOLOGY PROGRAM (NASA. Lewis Research Center) 17 p CSCL 20D
N89-26174
Unclas
G3/34 0217634

Robert J. Simoneau
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio

and

Dale A. Hudson
U.S. Air Force
Wright Research and Development Center
Wright Patterson AFB, Ohio

Prepared for the
25th Joint Propulsion Conference
sponsored by the AIAA, ASME, SAE, and ASEE
Monterey, California, July 10-12, 1989

CFD IN THE CONTEXT OF IHPTET--THE INTEGRATED HIGH PERFORMANCE TURBINE ENGINE TECHNOLOGY PROGRAM

Robert J. Simoneau
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

and

Dale A. Hudson
U.S. Air Force
Wright Research and Development Center
Wright Patterson AFB, Ohio 45433

SUMMARY

The Integrated High Performance Turbine Engine Technology (IHPTET) Program is an integrated DOD/NASA technology program design to double the performance capability of today's most advanced military turbine engines as we enter the 21st century. Computational fluid dynamics (CFD) is expected to play an important role in the design/analysis of specific configurations within this complex machine. In order to do this, a plan is being developed to ensure the timely impact of CFD on IHPTET. This paper, which will introduce a panel discussion on the title subject, is part of that process. The paper describes the developing philosophy of CFD in the context of IHPTET. It describes the key elements in the developing plan. The paper includes specific examples of state-of-the-art CFD efforts which are IHPTET turbine engine relevant. It attempts to make an evaluation of progress to the goal and to set some direction to the future.

INTRODUCTION

The Integrated High Performance Turbine Engine Technology (IHPTET Program is an integrated Air Force, Army, Navy, DARPA, and NASA technology program designed to double the performance capability of today's most advanced military turbine engines as we enter the 21st century. Doubling capability means different things in different applications. As appropriate to the specific application it could mean increasing thrust-to-weight, decreasing specific fuel consumption, increasing power-to-weight, and so forth. No matter what the criteria, the demands bring the goals close to the ultimate turbine engine. All of the technology tools will be stressed to their limits.

The role and impact of computational fluid dynamics (CFD) has been the subject of considerable discussion among the IHPTET technology planners. Those planning component technology and engine demonstrator work recognize the importance of advanced analytic capability in helping designers make critical and timely decisions need to meet program goals. A central question is, however, whether and to what extent CFD can rise to the challenge of being useful, accurate, reliable, and timely in order to significantly impact IHPTET technology.

The challenge has been expressed by the co-chairman of the IHPTET steering committee, Dr. Donald Dix, on numerous occasions, one which was the Aeropropulsion '87 Conference at NASA Lewis Research Center. At that time, November 1987, Dr. Dix set out his thoughts on CFD in the context of IHPTET.

- The need: Necessary for timing of IHPTET goals

- The fact: No perfect solutions foreseeable

- The challenge: Configuration-specific, imperfect solutions, useful for design/analysis purposes

In September 1987, an interagency IHPTET CFD planning team was formed and added to the IHPTET steering committee structure. The authors of the present paper are chairman and member, respectively, of that team. The committe was charged with the responsibility of identifying needed IHPTET-related CFD, developing CFD plans or roadmaps, and integrating them into the component roadmaps. The group was also to establish long term working relationships.

This paper is an update on the progress in completing these tasks. It examines the philosophy of bringing a fundamental discipline to bear on an applied technology program. It outlines a framework for building a plan and draws on some specfic examples to illustrate IHPTET-relevant CFD. Some assessments and projections for the future are included. We being with some thoughts on the DOD and NASA Lewis perspectives to set the stage.

DOD PERSPECTIVE ON CFD

The IHPTET initiative has challenged both government and industry organizations to identify those technologies which are critical to meeting a highly aggressive set of goals. Across the industry three principle technology areas have been identified: (1) computational fluid dynamics (CFD), (2) advanced structural designs, and (3) advanced materials. Clearly, CFD is a key player in our efforts to double propulsion capability as we enter the 21st century. The CFD efforts in this country can easily be divided into two categories: (1) the highly theoretical, university level development of physical models and numerics, and (2) highly applied design and analysis of complex physical hardware. DOD has chosen to concentrate its CFD efforts on supporting application to design and experimental support of hardware development. This choice has allowed DOD to tailor its support of CFD to match the needs of the materials or component under development. Most of the advanced materials require highly controlled thermal or aerodynamic environments, environments far beyond the capability of classical development technology. The complexities of these new hardware components have also exceeded the capability of classical design development technology to achieve. DOD efforts to support CFD applications have highlighted two areas requiring increased effort: (1) the existing body of data to support development is totally inadequate in both detail and completeness and must be expanded; (2) the models themselves lack the rigorousness required to fully and accurately describe hardware flow fields. The impact of these limitations is increased design effort and testing. The response has been the expansion and increased sophistication of data collection required in development programs. CFD development is benefitting directly from this new data because it provides the quantity and quality of data needed by code developers. Both the research and the industrial communities receive synergistic benefits by the data used to generate CFD capability based upon configurations closely aligned with product needs. The quality of both CFD and gas generator hardware is being improved through the funding of CFD supporting data in our technology development programs. What has been left out of this discussion is, "Who does this physical model improvement?" DOD recognized that CFD development is a job not well suited to development under the type of time schedule to which IHPTET is being placed. Our best hope is to draw upon CFD models currently under development and nearing application. These available models could then be transitioned to analysis and design use. It is at this point where the IHPTET partnership with NASA has some of its greatest payoffs. NASA is well positioned for working with the basic research community while also helping to transition that technology to the industrial community.

NASA PERSPECTIVE ON CFD

CFD is s significant part of the NASA Research and Technology program. The CFD efforts at the reseach centers, NASA Lewis, Langley, and Ames, tend towards the development of physical models and numerics as well as modeling and validation experiments. NASA Lewis leads the aeropropulsion research and much of its CFD effort is directed towards the flow physics of the turbine engine. Part of the work is done in-house at NASA Lewis and thus provides a technical cadre of experienced professionals as well as the CFD output.

NASA is pleased to be a partner with the DOD on IHPTET, especially in FD. NASA can make significant specific contributions as suggested in the revious section. Also the partnership provides an important focus to the esearch and the DOD effort provides a real component/engine environment chnology base to support and test the predictive capability of the mputational tools.

IHPTET CFD PLAN

It should be emphasized right from the start that the IHPTET CFD plan is ll evolving. In fact, it is a primary purpose of this paper and the com-ion panel[1] discussion to evoke discussion and response to the developing l. On the other hand, considerable discussion has already gone into the and this seems to be an appropriate time and an appropriate forum to ew its framework.

---

[1] Panel on CFD in the Context of IHPTET
G. Pickett, Pratt & Whitney, East Hartford, CT; CFD Contributions in Gas ne Design.
l. Mongia, General Motors Corp., Indianapolis, IN; Assembling CFD Tools n Effective Design/Analysis Scheme.
. Suo, GE Aircraft Engines, Cincinnati, OH; Connecting the Code er to the Designer.
Strazisar, NASA Lewis Research Center, Cleveland, OH; Experimental rations of Component Analysis Capability.

It is also important to realize that the plan is a slice in time. It has a past as well as a future. While it is the future that concerns us, it is important to note that CFD is already impacting turbine engine design/analysis in a significant way. This will be discussed more in the next section.

The challenge outlined in the introduction is really the challenge heard throughout the turbine engine community. For CFD to impact IHPTET, it must be configuration-specific, it must be useful for design/analysis purposes, it must capture/identify physics key to the design of the component being analyzed, it must have demonstrated capability, and it finally must be timely.

Looking first at the key physics question, the committee attempted to identify important physical phenomena requiring analyses and the CFD needs to do the job. These are illustrated in figure 1. The list is not intended to be exhaustive, but certainly gives the flavor and challenge of the task. In making such a list, it is important to recognize that a code does not have to be perfect to be useful--to capture key physics. For example, a full three-dimensional Navier-Stokes code may still need considerable work to resolve viscous losses in turbomachinery blade rows, but may do a wholly adequate job of predicting secondary flow patterns and/or three-dimensional shock structure. If it can be demonstrated that it can do the later, then it should be used and should not wait for perfection. However, caution must be exercised that codes are not used beyond their demonstrated capability lest they lead the designer astray, and in the process lose their credibility.

This leads to two of the three main elements of the plan; the first discussed above, is being satisfied that all key physical phenomena requiring analyses are receiving attention (fig. 1). The next two elements are the concepts of component analysis capability demonstrations and of establishing a path of credibility.

Many CFD groups, certainly NASA Lewis, have adopted the position that CFD extends well beyond code development. Code development is seen as one leg of an important triangle, which includes physical modeling and also exeriments to get at the physics and validate the codes. This is illustrated in figure 2 with added dimensions important to IHPTET. First, the CFD loop must be IHPTET specific. The code development, the physical modeling, and the experiments must be for geometries and physics one would expect to find in a modern turbine. Second, the codes need to have their predictive capability (for the key physics in question) verified in component test rigs. These experiments can both offer the challenge and provide the data which can resolve whether the challenge was met. These will normally be highly instrumented component test rigs which offer realistic geometries and operating conditions, while at the same time providing good control and definition of boundary conditions as well as detailed data. The key to success will be in identifying good experiments that can test the codes in a timely manner and in sharing the results with the IHPTET community. An example of such experiments are the transonic fan work of Strazisar (ref. 1), Pierzga and Wood (ref. 2) at NASA Lewis and Wennerstrom (ref. 3) at WRDC.

The final key point to the plan is attempt to establish paths of credibility that the IHPTET-related CFD will flow into the component/demonstrator program in a reliable and timely manner. To illustrate this, we have selected one program element in the IHPTET CFD plan: the multistage flow physics program. (Actually, there are two multistage program elements; one in compressors and one in turbines.) This program is based on the average passage ananlysis codes of Adamczyk and co-workers (refs. 4 and 5). Consistent with the model in figure 2, this program is not only the development of a code. It includes: (1) extensive model development, (2) experiments in a large, low speed axial compressor to aid in the modeling and calibrate the code, (3) a planned contract in a turbine rig to get at stage effects on secondary flow, and (4) developments in parallel processing to speed the calculations and to enhance the data acquisition and analysis. (It should be noted that there are other CFD efforts nationally directed at multistage physics. This was chosen because it is on the NASA Lewis IHPTET-related CFD roadmaps and will be in the public domain.) The flow of this program element in the context of IHPTET is shown in figure 3.

A few observations are in order. First, the path shown by the heavy arrows is an ideal path where a new level of confidence is established at each step. Other more direct paths can and will occur. Some possible alternate paths are shown in dashed lines. They must, however, be used with caution and understanding. Furthermore, the flow is not one way. There is considerable feedback and corresponding improvement. In order to avoid overloading the chart, not all paths are illustrated. The main point is to suggest that it is possible for advanced CFD tools, such as the multistage physics program/average passage analysis, to flow into IHPTET component design/analyses in a crediable and timely manner. It is very reasonable to expect that the program chosen for discussion can impact Phase II IHPTET multistage machines. For the IHPTET CFD plan to be successful, all of the program elements will have to be reviewed against a model such as illustrated in figure 3, for their timely impact on the IHPTET roadmaps. It should be noted, as illustrated in figure 3, that there is considerable overlap at each level. There is not just one build or one entry point. The tools can, in fact, enter over a several year interval to have a timely impact. On the other hand, there is a specific window, and resources will have to be properly aligned to match the opportunities.

The detailed roadmaps, showing IHPTET-related CFD program elements, such as the one discussed above, are under development and will not be discussed in detail herein. They will have elements in each of the component areas: (1) fans/compressors, (2) combustors/augmentors, (3) turbines, and (4) nozzles. Many of the near-term (next 5 years) programs are in place. They tend to emphasize the time-averaged physics and are expected to impact Phase II of IHPTET. The Phase III planning is less developed, but will probably emphasize time-resolution and multidiscipline/multisystem analyses. Before looking to the future, it is appropriate to first look at where we stand now.

STATUS OF TURBINE ENEGINE CFD

The key physical phenomena of CFD for turbine engines are summarized in figures 1(a) and (b). However, these needs must be put in an IHPTET contect so that priorities can be established and so near term component and engine design efforts can be helped.

The critical needs of component and engine desingers address the accurate knowledge of physical, arerothermal processes as influenced by flowpath hardware. The use of CFD can be grouped into three categories: (1) routine design, (2) developmental problem solving or design checking, and (3) detailed analysis. Each level is characterized by increasing complexity, detail, and difficulty. At the "routine design" stage, CFD is predominately two-dimensional, inviscid and boundary layer codes. The "problem solving" or "design checking" level is dominated by Euler Solver codes. The "analysis" level is primarily composed of three-dimensional boundary layer and Navier Stokes codes, both steady and unsteady. Each level provides the designer with new data that confirms or rejects design choices.

The basic parameters of bulk flow velocities and pressure fields are routinely solved by two- and three-dimensional potential field, boundary layer, and Euler Solver codes. These codes are patched together to provide the required solutions. It is in the "design checking" or "problem solving" phase in which today's "design" state-of-the-art is defined. Both compressor and turbine designers now use Euler Solver codes to refine hardware configurations. Many of these Euler codes produce steady state solutions to multiblade row solution fields. Just as with the "routine" code solutions, these intermediate codes are patched together to give complete flow field solutions. At this level of refinement boundary treatments become a fundamental part of the codes accuracy. Body fitted coordinates, whether orthogonal or not, are vital. The unique nature of combustion has already driven the combustor designer to three-dimensional viscous flow codes. The TEACH type two- and three-dimensional viscous codes are by far the prevailing tools. Chemistry and radiation models continue to be the weakest parts of these codes.

At this point, we move beyond the routine to the CFD codes that the designer frequently sees as the ones of last resort, those physical models, algorithms and solution processes still under development. For discussion, here the CFD needs of a designer are divided into rotating machinery needs and needs not implicitly tied to rotating hardware. For rotating machinery the needs are shock and boundary layer interaction, viscous/inviscid interactions, three-dimensional boundary layer, and blade row interactions or unsteady aerodynamics and, finally, heat transfer. Between the compressor and turbine, the fluid temperature and composition may be slightly different and the pressure field reversed but, except for heat transfer, the key physical phenomena listed above are the state-of-the-art problems in all rotating turbomachinery CFD. Not implicitly tied to rotating hardware are viscosity effects, chemical reaction, heat release, combustion unsteadiness, and both steady and unsteady heat transfer. It is in this final level that the complete flowpath solution is coming from one code. The ever present need for improved turbulence modeling goes with all classes of needs. Of somewhat growing importance is large scale turbulence. A growing body of data is pointing at large scale turbulence as a key contributor to unsteady flow influence on hardware performance. These needs are not brand new and both NASA and DOD have efforts underway which are providing important contributions in these many areas.

Experimental support for CFD is also underway at both NASA and within DOD. However, in the development of CFD these two partners take significantly different paths. As mentioned earlier in the perspectives, the DOD supported work stays closer to the design and the hardware, although in practice one will find many similarities in approach. Nowhere is DOD's support of engine design and development aspects of CFD seen better than in current work efforts. The NAFCOT Fan, developed by Pratt & Whitney Aircraft under a joint Navy/Air Force contract, shown in figure 4, was tested in WRDC's Compressor Research Facility. This program is an outstanding example of advanced technology providing critical data never before available. Advanced laser instrumentation was used to map the two- and three-dimensional flow field of this highly swept fan during full speed operation at design conditions. The spatial matrix of data sampling points is illustated in figure 5.

Theoretical analysis is not being totally left to NASA Lewis as evidenced by the Air Force's recent initiation of an in house effort to develop a three-dimensional, quasi steady, Navier-Stokes code for analysis of 2-1/2 stages or five blade rows of a compressor. The solution grid and pressure field of one of these blade rows are illustrated in figures 6 and 7. Under this effort the Air Force hopes to accurately capture shock wave behavior through successive blade rows. Similarly, the turbine area is actively pursuing unsteady flow effects, including shocks, through multiple blade rows. A good example of this effort is the Air Force Vane Blade Interaction program with Allison Gas Turbine Division of General Motors Corporation. The program goal is to develop the analysis capability to capture shocks, unsteady aero effects and heat transfer. To date, work has focused on a two-dimensional Euler Solver to establish the rotor and stator interaction. Stator and rotor solutions are iterated to satisfy interboundary conditions at grid overlap points. The grid and pressure field are shown in figures 8 and 9. The analysis code accurately reproduces major flow features including shocks. Remaining work has two options, to extend the code to a full three-dimensional Euler Solver or to pursue a Navier Stokes solution so that viscous effects can be picked up and heat transfer coefficients can be evaluated. Ideally, an unsteady blade to blade Navier Stokes analysis capability would capture all the major flow features, including shocks and wakes and their effect on heat transfer. The importance of such a solution is emphasized by current estimates that wakes from blades cause a factor of five increase in heat transfer coefficient on the surface of following blades. Experimental data directly supporting the Vane Blade Interaction program is being conducted in the Calspan transient turbine rig. The experimental stator and rotor blade rows are full scale and allow direct comparison between analysis code predictions and experimental data. The experimental results of this research have been periodically reported at technical conferences by Dunn et al. (ref. 6). In the combustor area, the Air Force and NASA are currently conducting a joint effort on the diffuser between the compressor and combustor. The Combustor Diffuser Interaction program, being conducted by Garrett Engine Division of Allied Signal Aerospace Company, is developing a higly detailed and comprehensive data set documenting flow in a realistic test configuration. The experimental apparatus is shown in figure 10. A diagram of the test configuration, with representative data at measurement plane locations, is given in figure 11. When completed this program will provide a comprehensive data set useful for developing and validating advanced design codes. The effort will also produce a new combustor diffuser design which will be the basis for Garrett's future engine systems.

Most of the programs discussed above are closely tied to current hardware development efforts. All are directly focused at providing better design and analysis capability which will impact the IHPTET initiative. The DOD and NASA efforts to coordinate CFD and supporting programs are taking advantage of these programs and others like them to provide the critical "path of credibility" for CFD from basic research environment to the design shop where tomorrow's engine systems take shape.

IMPACT OF CFD ON IHPTET

The grand goal of doubling performance capability cascades into many individual perfromance goals for each component--temperature, pressure, pressure ratio per stage, work per stage, efficiency, weight, etc. Each of these component technology goals has a set of enabling technologies, for example: achieving a certain material temperature. Three-dimensional steady and unsteady viscous CFD is cited in several areas as one of these component technology enabling technologies.

The question for the IHPTET planners is how the enabling technology of three-dimensional viscous steady/unsteady CFD will impact IHPTET. First, it must be noted tha computational design/analyses have been impacting turbine engine design for years. So when we ask the question of impact, we must be sure we are on the same foundation; that the foundation being the full three-dimensional viscous analyses. The impact of these analyses will depend on the accuracy, reliability, fidelity, and speed with which the computational analyses reproduce the critical physics of turbine engine components.

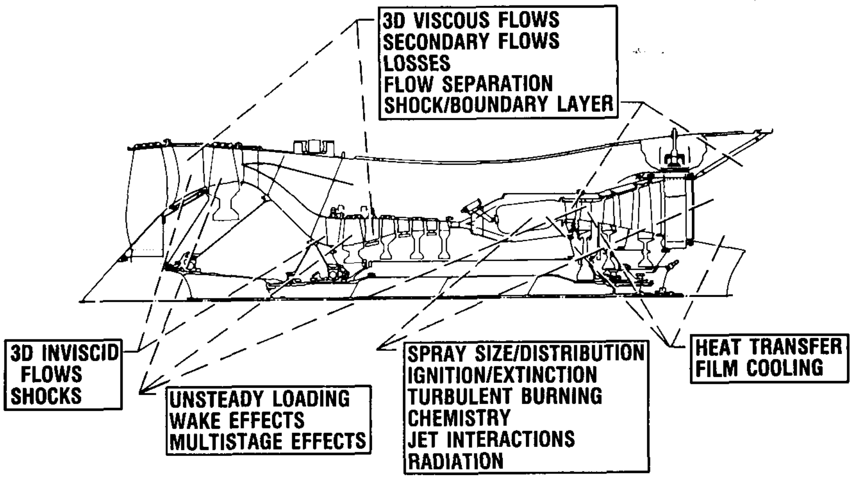
In order to meet that test, CFD has its own enabling technologies--one level further in the refinement of IHPTET goals. In figure 12, an attempt is made to examine the enabling CFD technologies against the IHPTET timetable. In Phase I, which is almost history as far as CFD in IHPTET is concerned, the emphasis was on numerics and building codes to take advantage of the explosion in computer hardware. In Phase II, which for CFD is on us now, the growth in algorithms will continue, but there will be a strong shift towards modeling the physics that cannot be captured by brute force numerics. We already have excellent computational tools and they are growing steadily. The Phase II interval will see a combination of advanced codes, advanced configuration-specific experiments, and physical modeling, such as described in the discussion of figures 2 and 3, to bring reality and reliability to the design/analyses systems. As Phase III is approached, these will all continue and a new explosion in computing--massive parallel processing--should allow growth to multidiscipline, multicomponent analyses, approaching an interactive design system.

If the CFD community (which includes code developers, modelers, and experimentalists) can develop the enabling technologies in a timely manner, then in the opinion of the authors the impact of CFD on IHPTET can be summarized as follows. First, in Phase I, the impact of CFD has been largely after the fact. CFD has been used to examine new design frequently after they have been built. The analyses have been very useful in exploring the reasons for good or bad performance and suggesting improvements. It is a new tool--one of many--gaining acceptance and use, but with many skeptics. Phase II will see CFD impacting IHPTET concurrently with the design. In an interactive fashion using the codes, the data, and the models, the designer and the analyst will work together to improve design before the metal is cut--or the ceramic is fired. By Phase III, the prosepct is for CFD to lead the design, not completely, but in select areas where the details of some local physical phenomenon could affect the overall performance--or vice-versa--the overall performance could affect some important physical detail.

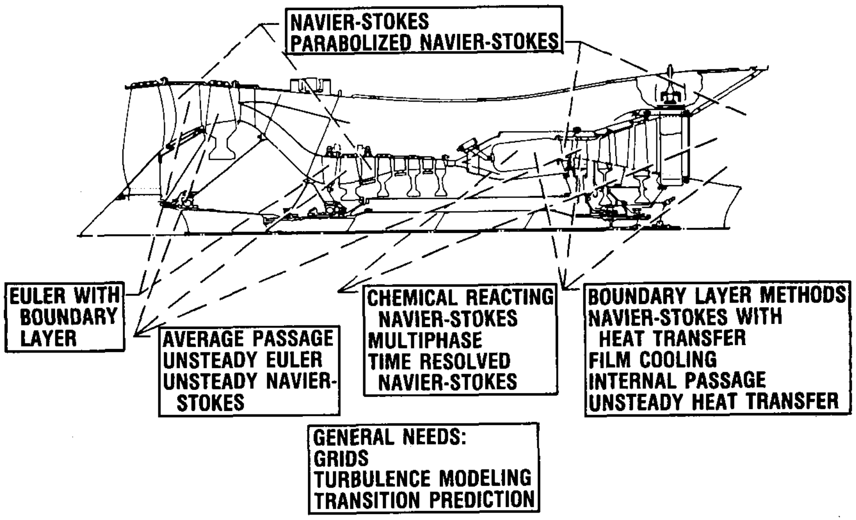
Because this type of capability requires long lead time and considerable advanced planning, a model for this is already under development at NASA Lewis (ref. 7). This model is illustrated in figure 13 and is being described as

6. Dunn, M.G., Seymour, P.J., Woodward, S.H., George, W.K. and Chupp, R.E.; "Phase Resolved Heat-Flux Measurements on the Blade of a Full-Scale Rotating Turbine," ASME Paper 88-GT-173, June 1988.

7. Miller, B.A., Szuch, J.R., Gaugler, R.E. and Wood, J.R., "A Perspective on Future Directions in Aerospace Propulsion System Simulation," paper presented at the Fourth International Conference on Supercomputing, Santa Clara, CA, May 1989. (NASA TM-102038).



(a) PHYSICAL PHENOMENA REQUIRING ANALYSES.



(b) CFD NEEDS.

FIGURE 1. - CONCEPTUAL ADVANCED TURBOFAN ENGINE.

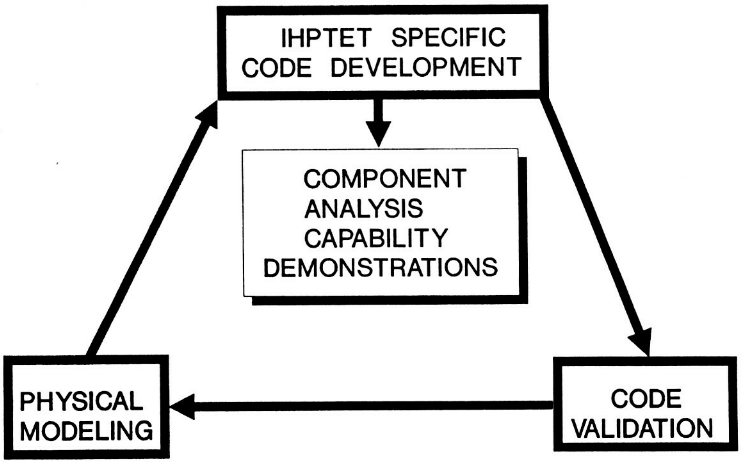


FIGURE 2. - CFD IN THE CONTEXT OF IHPTET.

PATH OF CREDIBILITY

MULTI-STAGE FLOW PHYSICS PROGRAM

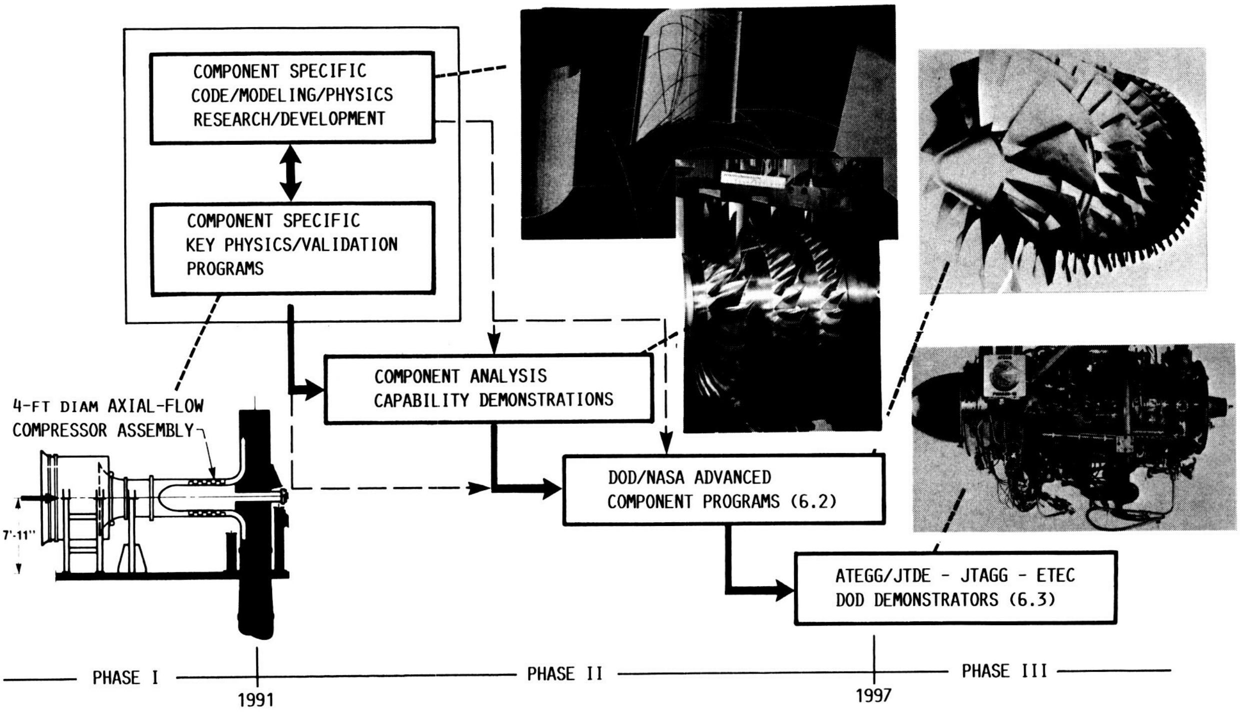


FIGURE 3. - CFD FLOW INTO IHPTET COMPONENT/DEMONSTRATOR PROGRAMS.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

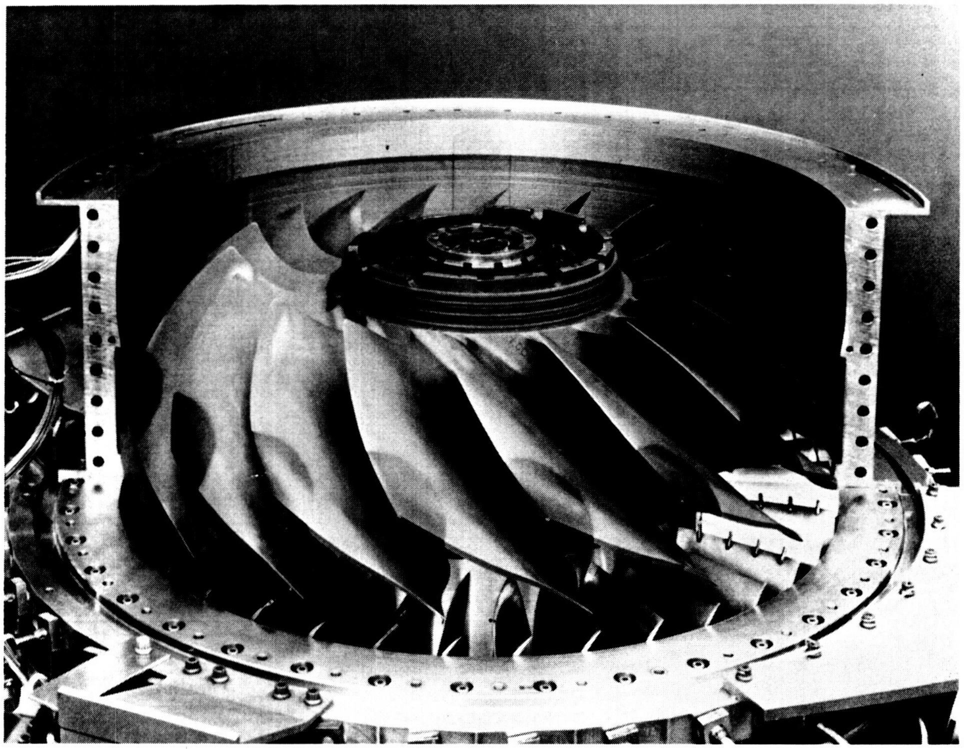


FIGURE 4. - NAVY ADVANCED FAN COMPRESSOR TEST.

NAFCOT LASER TEST

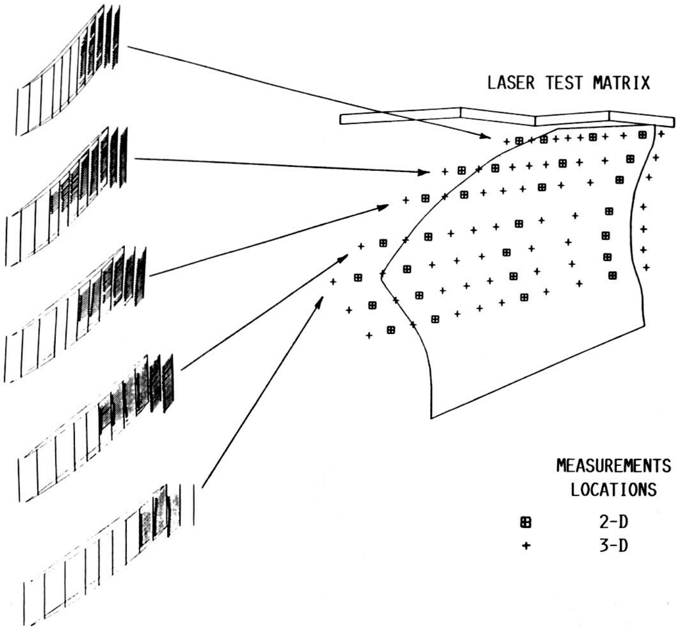


FIGURE 5. - SPACIAL TEST MATRIX.

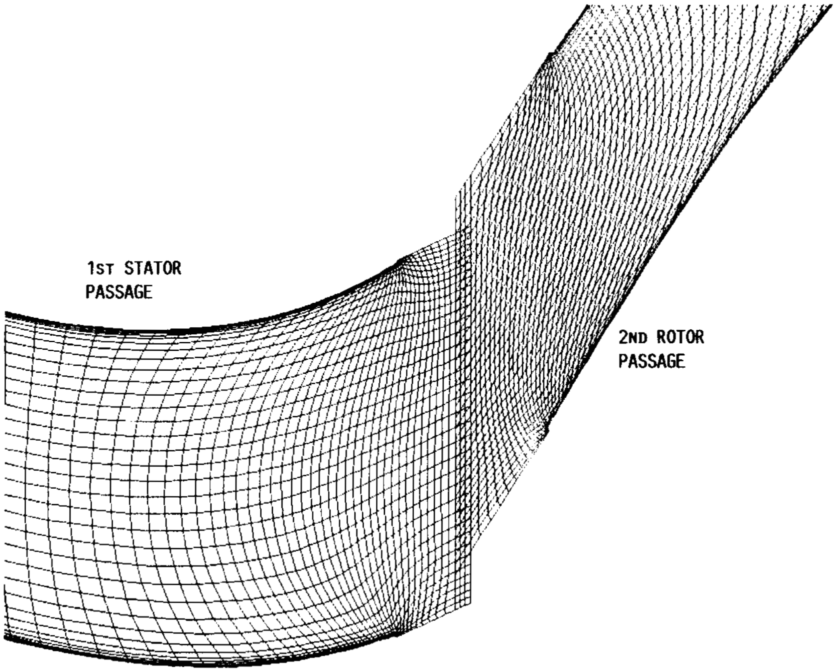


FIGURE 6. - OVERLAP REGION OF ROTOR/STATOR INTERACTION NAVIER-STOKES SOLVER GRID.

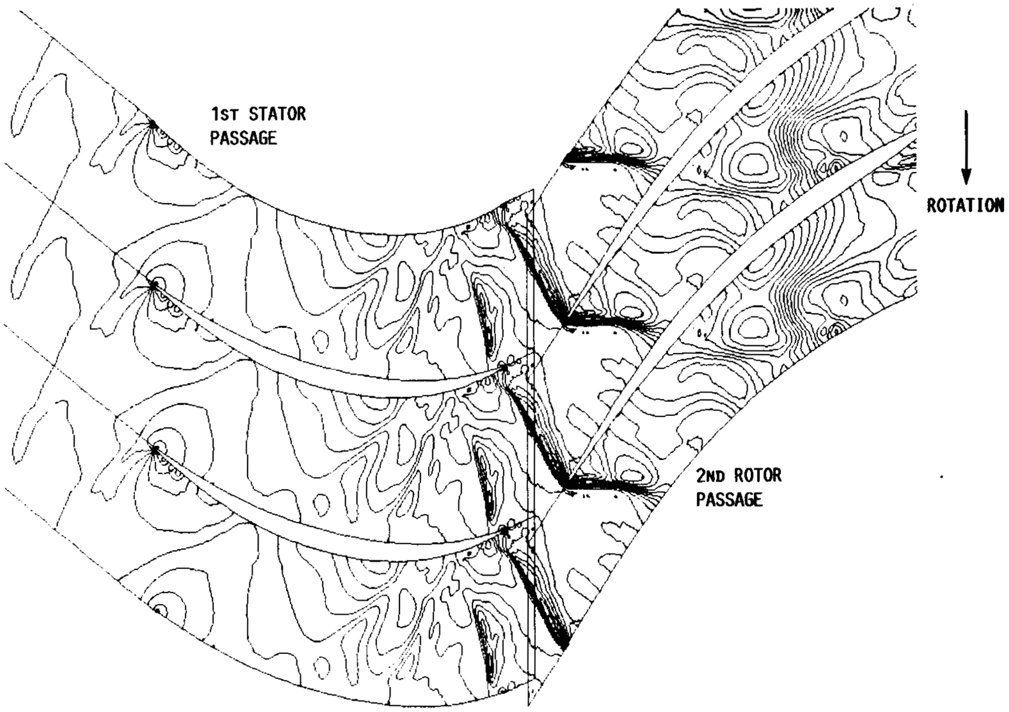


FIGURE 7. - NAVIER-STOKES CALCULATION OF BOW SHOCK-WAKE INTERACTION PRESSURE CONTOURS AT 90 PERCENT SPAN IN A 2-1/2 STAGE COMPRESSOR.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

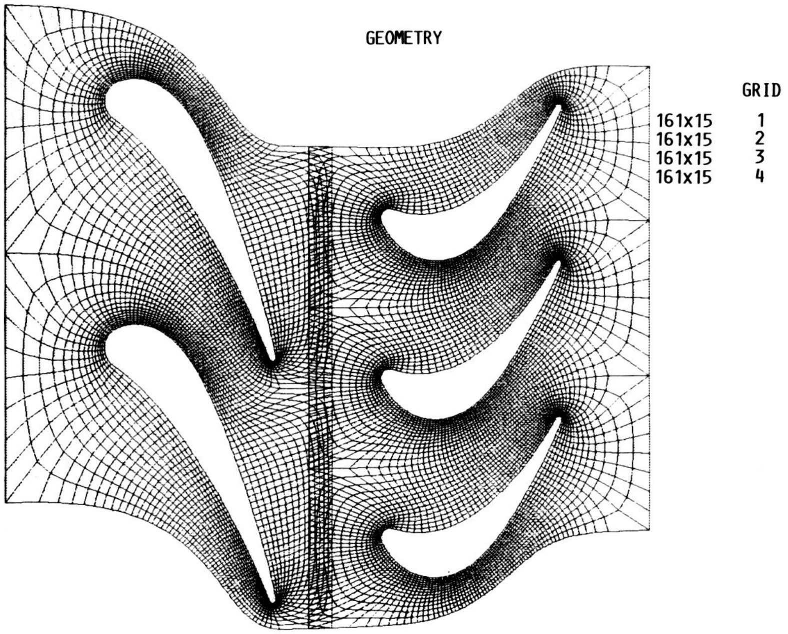


FIGURE 8. - VANE BLADE INTERACTION SOLUTION GRID.

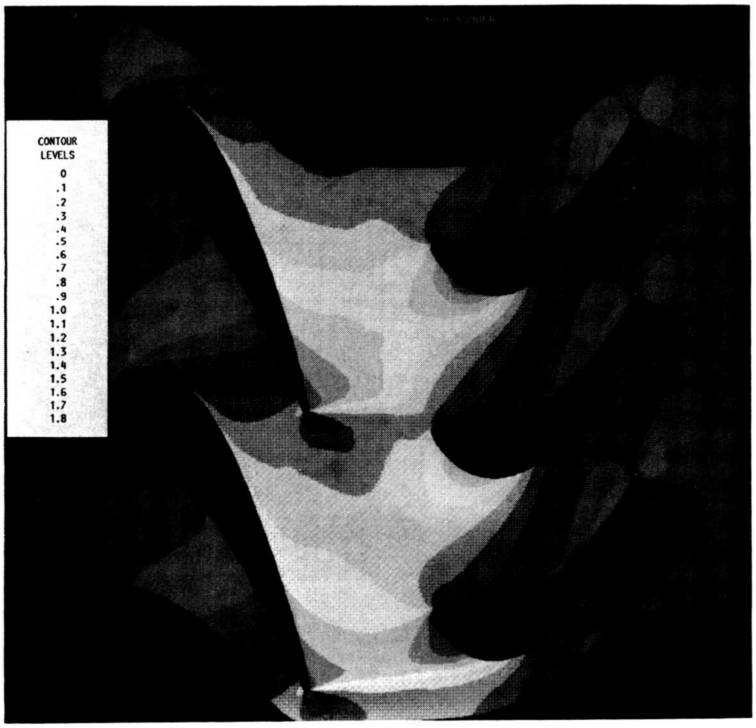


FIGURE 9. - VANE BLADE INTERATION MACH NUMBER FIELD.

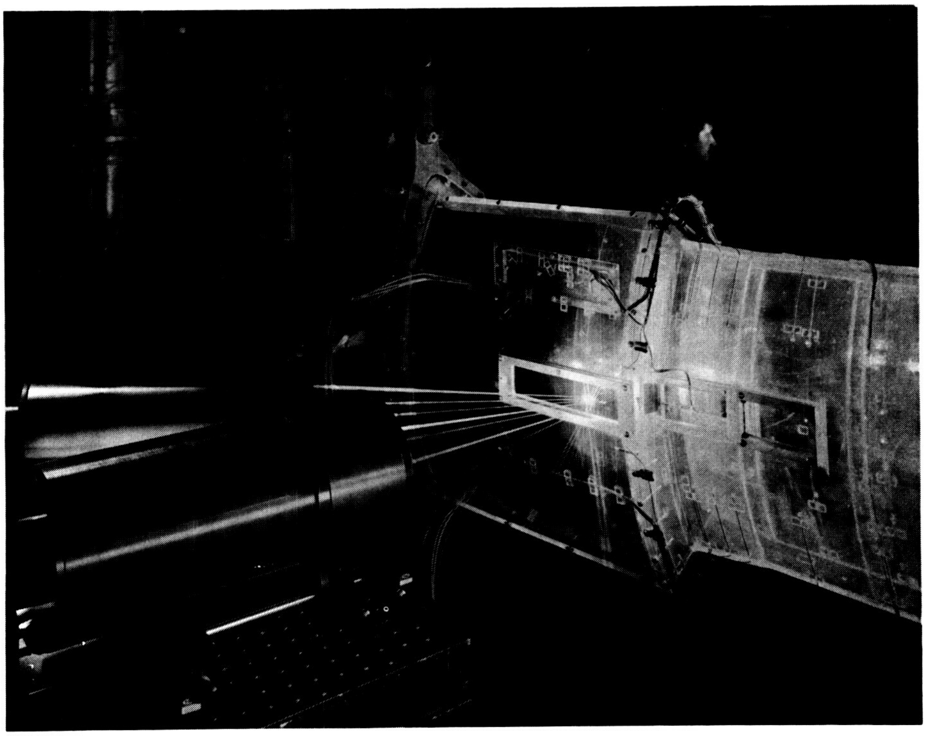


FIGURE 10. - COMBUSTOR DIFFUSER INTERACTION TEST RIG.

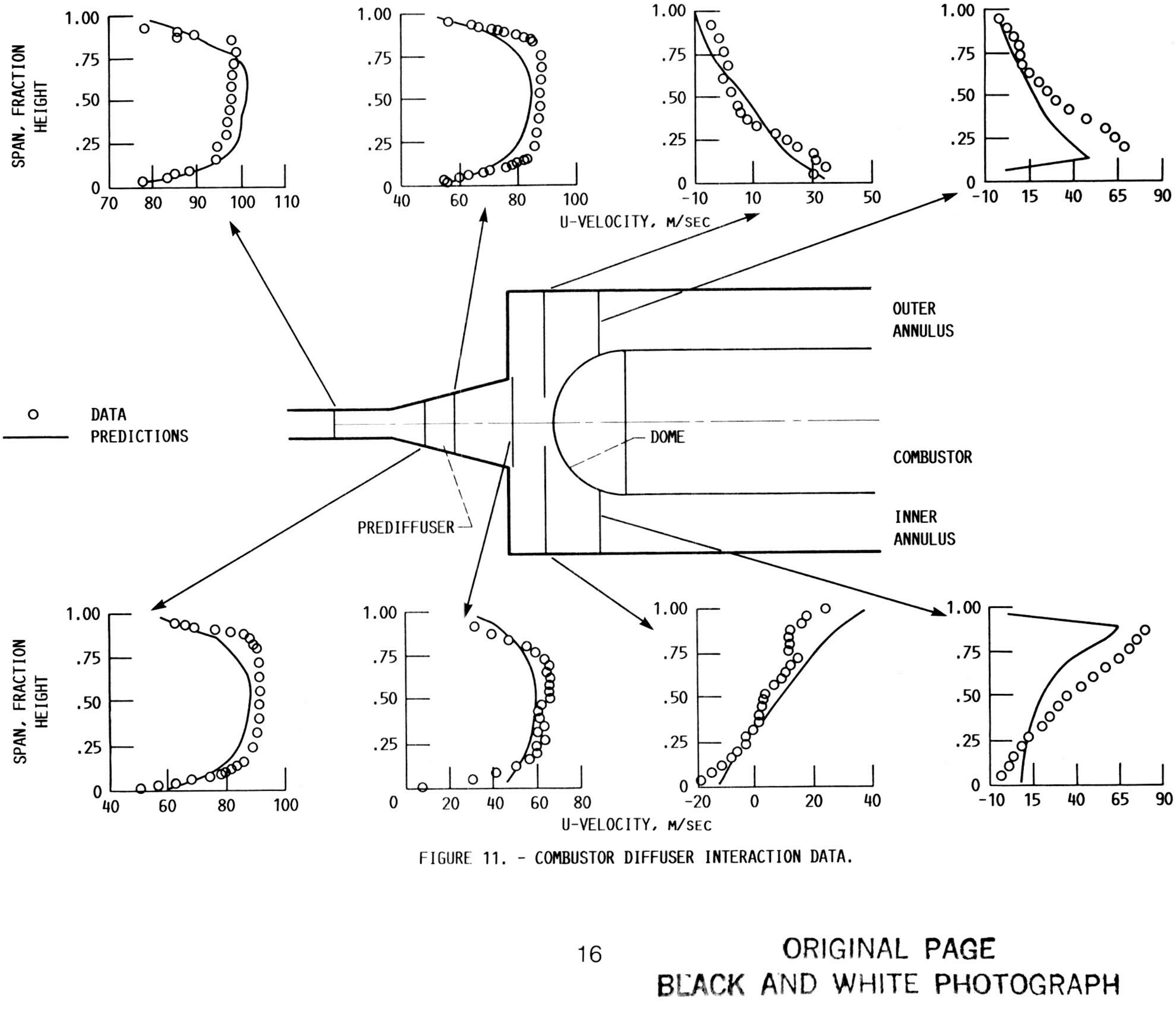


FIGURE 11. - COMBUSTOR DIFFUSER INTERACTION DATA.

Phase I, Gen 5 (1991)

- 2D/3D Euler
- Parabolized Navier-Stokes
- 2D/Quasi-3D Viscous
- Boundary Layer Heat Transfer
- k-e Turbulence Modeling
- Algorithms/Grids

CODES, NUMERICS

Phase II, Gen 6 (1997)

- 3D Viscous w/Heat Transfer
- 2D Unsteady
- Average Passage Sys Modeling
- Vast Time/Length Scale Range
- Real Finite Rate Chemistry
- Turbulence Modeling
- Transition Modeling

PHYSICAL MODELING

Phase III, Gen 7 (2003)

- Interactive Des/Anal
- Num Prop Simulation
- High Perform Compute
- 3D Unsteady w/H.T.
- Closure Modeling
- Turbulence Modeling

FULL SYSTEMS

FIGURE 12. - ENABLING CFD TECHNOLOGIES.

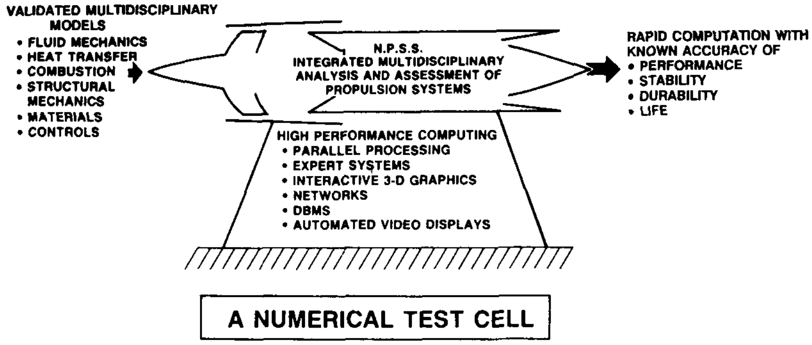


FIGURE 13. - NUMERICAL PROPULSION SIMULATION SYSTEM (N.P.S.S.).

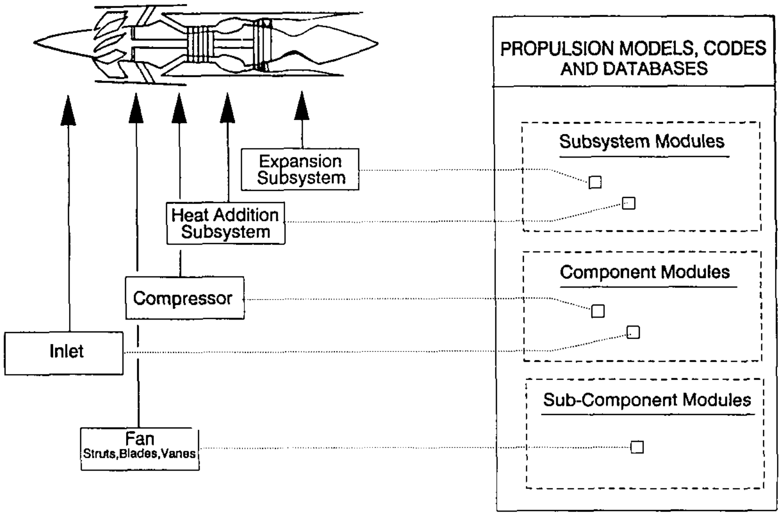


FIGURE 14. - NUMERICAL PROPULSION SIMULATION SYSTEM. "ZOOMING IN" ON FAN EFFECTS.

National Aeronautics and
Space Administration

Report Documentation Page

1. Report No. NASA TM-102132
AIAA-89-2904

2. Government Accession No.

3. Recipient's Catalog No.

4. Title and Subtitle
CFD in the Context of IHPTET—The Integrated High Performance Turbine Technology Program

5. Report Date

6. Performing Organization Code

7. Author(s)
Robert J. Simoneau and Dale A. Hudson

8. Performing Organization Report No.
E-4868

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

15. Supplementary Notes
Prepared for the 25th Joint Propulsion Conference cosponsored by the AIAA, ASME, SAE, and ASEE, Monterey, California, July 10-12, 1989. Robert J. Simoneau, NASA Lewis Research Center; Dale A. Hudson, U.S. Air Force, Wright Research and Development Center, Wright Patterson AFB, Ohio.

16. Abstract
The Integrated High Performance Turbine Engine Technology (IHPTET) Program is an integrated DOD/NASA technology program designed to double the performance capability of today's most advanced military turbine engines as we enter the twenty-first century. Computational Fluid Dynamics (CFD) is expected to play an important role in the design/analysis of specific configurations within this complex machine. In order to do this, a plan is being developed to ensure the timely impact of CFD on IHPTET. This paper, which will introduce a panel discussion on the title subject, is part of that process. The paper describes the developing philosophy of CFD in the context of IHPTET. It describes the key elements in the developing plan. The paper includes specific examples of state-of-the-art CFD efforts which are IHPTET turbine engine relevant. It attempts to make an evaluation of progress to the goal and to set some direction for the future.

17. Key Words (Suggested by Author(s))
Turbine engine
Computational fluid dynamics

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 18

22. Price*

*For sale by the National Technical Information Service, Springfield, Virginia 22161