N87-20268

COMBUSTION RESEARCH IN THE INTERNAL FLUID MECHANICS DIVISION

Edward J. Mularz
Propulsion Directorate
U.S. Army Aviation Research and Technology Activity-AVSCOM
NASA Lewis Research Center
Cleveland, Ohio

At the NASA Lewis Research Center, combustion research is being conducted in the Internal Fluid Mechanics Division. The research is organized into three main functions focusing on the fluid dynamics related to aeropropulsion systems (fig. 1). The first function, computational methods, looks at improved algorithms and new computational fluid dynamic techniques to solve internal flow problems, including heat transfer and chemical reactions. This area also looks at using expert systems and parallel processing as they might be applied to solving internal flow problems.

The second function is fundamental experiments. These experiments can generate benchmark data in support of computational models and numerical codes, or they can focus on the physical phenomena of interest to obtain a better understanding of the physics or chemistry involved as a preamble to models and computer codes.

Computational applications is the third function in the Internal Fluid Mechanics Division. New flow codes are validated against available experimental data, and they are used as a tool to investigate performance of real engine hardware. Since the geometry may be quite complex for the system being analyzed, large grids and computer storage may be required. The hardware of interest includes combustion chambers, high-speed inlets, and turbomachinery components (e.g., a centrifugal compressor).

The goal of this research is to bring computational fluid dynamics to a state of practical application for the aircraft engine industry. As shown in figure 2, the approach is to have a strongly integrated computational and experimental program for all the disciplines associated with the gas turbine and other aeropropulsion systems by advancing the understanding of flow physics, heat transfer, and combustion processes. The computational and experimental research is integrated in the following way: the experiments that are performed provide an empirical data set so that physical models can be formulated to describe the processes that are occurring - for example, turbulence or chemical reaction. These experiments also form a data base for those who are doing code development by providing experimental data against which the codes can be verified and assessed. Models are generated as closure to some of the numerical codes, and they also provide physical insight for experiments. At the same time, codes which solve the complete Navier-Stokes equations can be used as a kind of numerical experiment from which far more extensive data can be obtained than ever could be obtained experimentally. This could provide physical insight into the complex processes that are taking place. These codes are also exercised against experimental data to assess the accuracy and applicability of models (e.g., the turbulence model). We feel that a fully integrated computational-experimental research program is more productive than other approaches and that it is the most desirable way of pursuing our goal.

Figure 3 is a cutaway view of a hypothetical combustor which illustrates the typical complex fluid mechanics and combustion features. The flows are highly three-dimensional with turbulence levels, in many cases, comparable in magnitude to the bulk velocity. Liquid fuels are injected as a spray which then undergoes vaporization and mixing. The chemical reaction which occurs causes changes in density and fluid mechanics properties and can cause the formation of a solid phase (soot) with its attendant high-radiation heat transfer. An understanding of these physical processes is needed before accurate numerical codes can be built and used as a predictive tool in the design process. In addition, the numerical methods for three-dimensional flows need improvements in accuracy and efficiency in order to properly simulate the features of these flows.

This then is the framework of the combustion research program. As shown in figure 4, the program is divided into four elements: advanced numerics, fuel sprays, fluid mixing, and radiation chemistry. The research in each of these elements is focused on the long-range objective of developing numerical codes that can be used as a predictive tool to describe both two- and three-dimensional flows. Once the codes can be used with confidence, industry will be able to integrate them into their combustor design system.

Before discussing the specific research currently taking place, I would like to describe the major computing hardware now being used at NASA Lewis to compute reacting internal flows. Lewis currently has a CRAY 1-S computer which is tied to an IBM 370 system for input/output. It has a relatively small storage capacity, but a very high calculation speed. In the near future a CRAY X-MP will be installed (fig. 5). It will replace the CRAY 1-S and will provide a modest increase in computer speed. Core storage, however, will increase dramatically from 2 million words to 36 million words, and a high-speed solid-state device, which is practically equivalent to core memory, will also be included. In addition, the new equipment will have two processors which will give Lewis some ability to start using algorithms that can take advantage of simultaneous processing and thus increase the computation speed even further.

In addition to the computers located at Lewis, NASA is building a "super computer" at its Ames Research Center called the Numerical Aerodynamic Simulator (NAS) (fig. 6). The NAS computer will be accessed at both NASA Langley and NASA Lewis through satellite linkage with a UNIX operating system. In its current configuration, the NAS computer is a CRAY 2 with 256 megawords of memory, which will enable it to make very large calculations of chemically reacting internal fluid flows. It will have four processors. In addition, as the state-of-the-art of computers advances changes will be made in the NAS computer to reflect these advances.

The current research in combustion can be summarized in six major activities. The first activity is called improved numerical methods for complex flows. Numerical error is a big problem with the current numerical codes. This numerical error must be reduced before the accuracy of physical models can be assessed. The grids used in calculations that are performed with state-of-the-art codes are very sensitive to the flow direction relative to that grid, and at high angles this sensitivity leads to serious numerical errors. Techniques to reduce numerical diffusion, or error, and to virtually eliminate it from calculations are being pursued. Work is also underway to establish second-order accurate closure models for turbulent reacting flow.

The second activity involves the development of techniques for making predictive calculations of chemically reacting flow. One of the more promising techniques, which is really in its infancy, is called direct numerical simulations, or DNS. In this technique, the Navier-Stokes equations are solved directly without any modeling of the turbulence. This technique is currently being applied to reacting shear layers. Although this is a relatively simple flow, there is nevertheless a lot of complexity associated with it. Much work remains to be done with this technique, but the results to date have been very promising.

Another technique that has been used in several kinds of flows is called the random vortex method. This technique accounts for the vorticity generated at the wall of the confined flow and solves the vorticity equation without any turbulence closure modeling. Calculated results of flow over a rearward facing step show many of the characteristics seen in high-speed movies of turbulent reacting flow experiments.

The third activity involves benchmark experiments for code development and verification. Two-phase flow research is currently underway. Detailed data are required for code assessment, and instruments which Lewis has helped to develop now show much promise of being able to make the appropriate measurements. (Those instruments are also being used to support icing research.) Research is also being conducted on numerical calculation of two-phase flow. With the data from the experiments, an assessment of the current code capability will be made to guide future code development efforts. In addition, an extensive set of experiments is being conducted to look at the mixing of dilution jets into a cross stream in a channel. A substantial range of parametric variables has been studied, and a very complete set of data has been established.

The next activity is computer code applications. Currently available codes are applied to real systems. The study of flow in the transition section of a reverse flow combustor is an example of such work. Here a flow in which fuel has already been burned has to undergo a 180° annular turn. Very strong secondary flows arise, and the analysis is very complex. Another example is the application of the Lewis-developed general chemical kinetics programs and chemical equilibrium codes to problems of practical interest. These codes are used throughout the world.

The next activity is code development for thermochemical properties and kinetic rates. Thermodynamic calculations for real gases are being performed in support of future combustion models. The modeling of chemical kinetic computations for complex reactions is also being actively developed in support of future computer codes.

The final activity in the current combustion research program is to advance the understanding of chemical mechanisms in reacting flow. Chemical kinetic rates are measured using a shock tube facility. The data from this experiment are used to assess the kinetic models for various fuels of interest in aeropropulsion systems. Detailed characteristics of one- and two-dimensional controlled flames are also being established. We are interested in studying the behavior of some of the minor species of flames and in looking at the soot nucleation growth and eventual soot consumption in flames. Radiation heat transfer is dominant in combustors and other aeropropulsion systems. The control of soot nucleation and growth is essential if radiation heat transfer is to be reduced. Also, some of the minor intermediates of combustion are being measured in an attempt to understand the detailed physics involved in flame fronts.

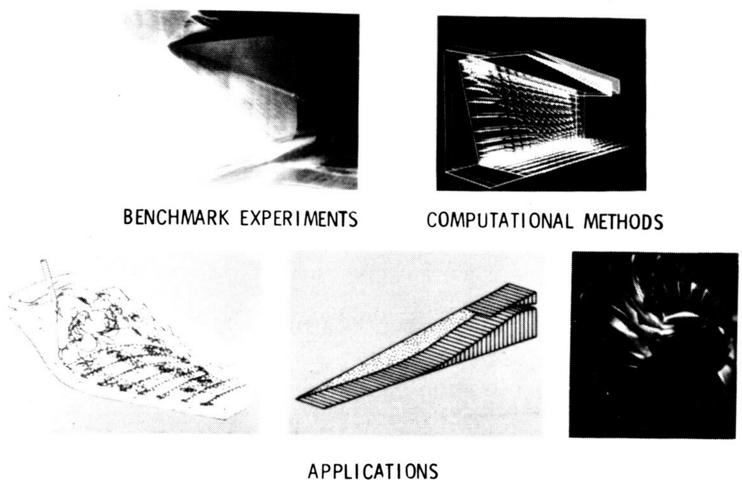

Figure 1. - Fundamental aeropropulsion computational and experimental research in the internal fluid mechanics division.

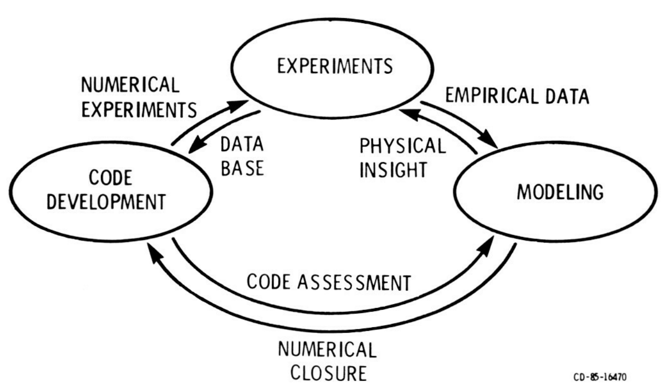

Figure 2. - Integrated computational-experimental methodology for propulsion aerothermodynamics research. Objective - advance the understanding of flow physics, heat transfer, and combustion processes which are fundamental to aeropropulsion.

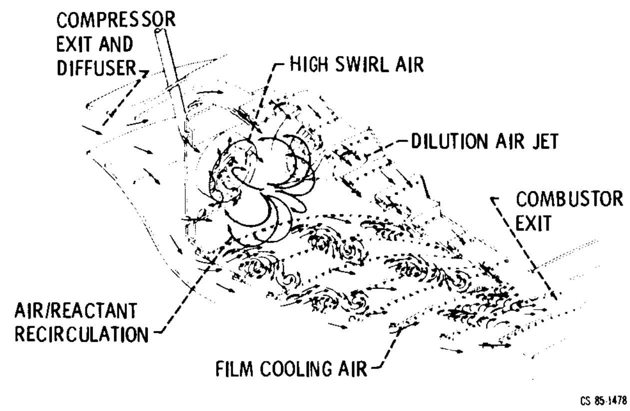

Figure 3. - Illustration of the typical flow phenomena in a gas turbine combustor. Typical flow is fully three-dimensional, has high turbulence levels, has chemical reactions and heat release, and occurs in two phases with vaporization.

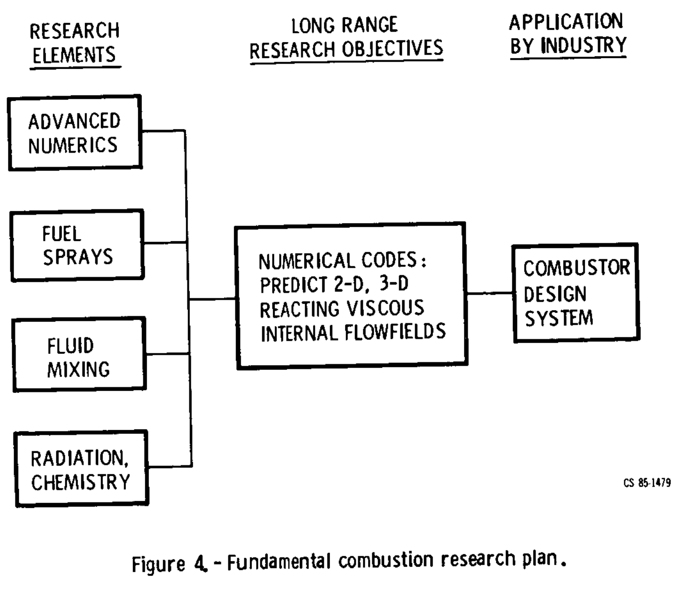

Figure 4. - Fundamental combustion research plan.

	CRAY 1-S (PRESENT)	CRAY X-MP (NOV. 85)
CYCLE TIME	12.5 nsec	9.5 nsec
STORAGE	2×10^6 WORDS	4×10^6 WORDS CORE 32×10^6 WORDS SSD
PROCESSORS	1	2

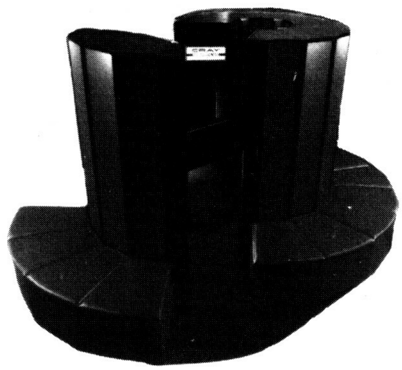

Figure 5. - Comparison of Cray 1-S and Cray X-MP computer capabilities.

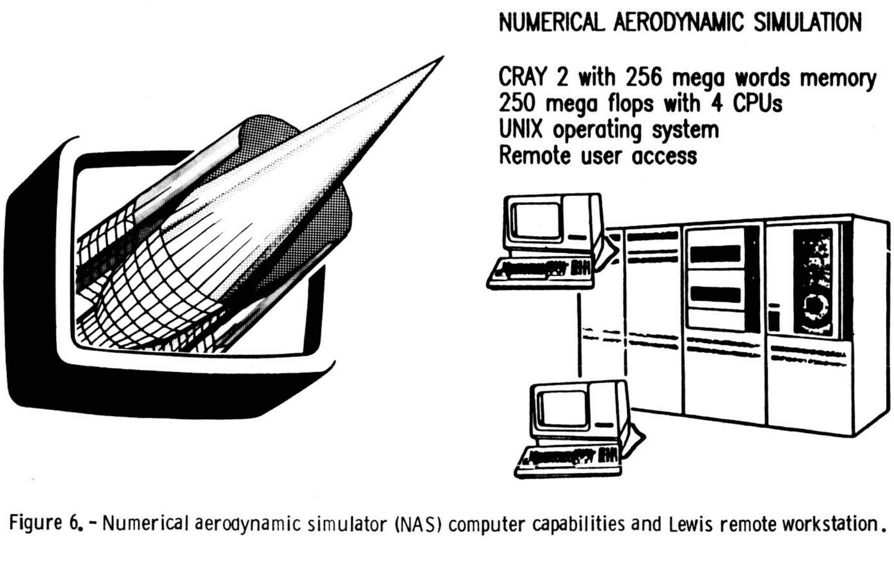

Figure 6. - Numerical aerodynamic simulator (NAS) computer capabilities and Lewis remote workstation.