Evolution and Application of CFD Techniques for Scramjet Engine Analysis

M. E. White*
The Johns Hopkins University Applied Physics Laboratory, Laurel, Maryland
and
J. P. Drummond† and A. Kumar‡
NASA Langley Research Center, Hampton, Virginia

Capabilities developed over the past several years in the area of computational fluid dynamics (CFD) have provided today's aerospace engineer with an extremely powerful tool for analysis of complex flowfields of interest. A recent resurgence in the interest in hypersonics has resulted in an increased effort in the development of supersonic combustion ramjet (scramjet) engines as an efficient means of propulsion for aircraft and missiles flying above Mach 5. Use of CFD techniques to design and analyze scramjet engine concepts is proving an invaluable asset, with applicability ranging from the use of simple inviscid codes for parametric studies of various inlet geometries to the use of unsteady Navier-Stokes codes with chemistry models to investigate the details of complex phenomena in a scramjet combustor. This article discusses the evolution of computational fluid dynamics with reference to milestones in scramjet research in the United States, followed by an overview of some applications of available computational capabilities to the design and analysis of scramjet engines being developed at the Johns Hopkins University Applied Physics Laboratory and at the NASA Langley Research Center. Finally, future directions in the application of CFD to the development of scramjet engines are discussed.

Introduction

COMPUTATIONAL fluid dynamics (CFD) has emerged as an extremely valuable engineering tool in aerodynamic design and analysis, due in part to the extremely rapid growth in the speed and storage capabilities of the digital computer. This rapid growth has been coupled with major advances in the development of numerical algorithms for the solution of the governing equations of fluid motion. Capabilities in CFD have been developed over the past 15 years for a wide variety of applications ranging from the solution of inviscid incompressible flowfields to the solution of the Navier-Stokes equations for high Reynolds number and Mach number. Until recently, a vast majority of the activity in the field has been focused on research to develop stable, accurate, and efficient numerical algorithms for solution of the equations governing the various flow regimes of interest. Although this much-needed research activity is continuing, the maturity of many computational techniques, as well as the growing availability of high-speed digital computers, has permitted the emergence of CFD as a design and analysis tool for today's engineer.

An area of particular interest in which a very powerful computational capability has been developed is the application of CFD to high-speed compressible flows. One such application has been motivated by the growing interest in the hypersonic flight regime,[1] which has focused considerable attention on the supersonic combustion ramjet (scramjet) engine for propulsion of both hypersonic aircraft and missiles. The recent progress in the application of CFD techniques has opened the way for a dramatic new approach in the design and analysis of scramjet engines. Computational codes for the solution of the supersonic flow through scramjet inlets are now in routine use. Although significantly more complex, computational codes have also been developed for the solution of the viscous, chemically reacting flow in scramjet combustors and nozzles. The emerging picture for the future of scramjet development shows a great dependence on the design and analysis tools made available by developments in computational fluid dynamics.

The objective of this article is to discuss several major topics in the application of CFD techniques to scramjet engines. First, the reader is provided with a cursory review of the evolution of CFD techniques, using as a reference point milestones in scramjet engine research and development in the United States. Important applications of CFD to scramjet analysis are emphasized. A discussion of some available computational capabilities and a methodology for application of these capabilities to scramjet engine design and analysis are then presented. This is followed by an overview of recent efforts in the use of CFD for design and analysis of scramjet engines being developed at the Applied Physics Laboratory and at the NASA Langley Research Center. Finally, the authors' viewpoint regarding future directions in the application of CFD to the development of scramjet engines is discussed.

Evolution of CFD—A Scramjet Perspective

The origins of both computational fluid dynamics and research in scramjet propulsion can be traced back to the late 1950s; however, their development remained independent until the 1970s, and it has been only recently that computational techniques have reached a level of maturity to become practical tools for the design and analysis of scramjet engines. In this section of the paper, an attempt is made to present the reader with some important milestones in the evolution of CFD capabilities and scramjet development efforts, with emphasis on applications of CFD to scramjets. The material presented is in no way intended to be all-encompassing but rather is designed to give the reader a sense of what has led up to the present technologies. For a

---

Presented as Paper 86-0160 at the AIAA 24th Aerospace Sciences Meeting, Reno, NV, Jan. 6-9, 1986; received Jan. 30, 1986.

*Aerospace/Project Engineer, Aeronautics Department. Member AIAA.

†Research Scientist, Computational Methods Branch. Member AIAA.

‡Senior Research Scientist, Computational Methods Branch. Member AIAA.

more detailed discussion on the development of scramjet engines in the United States, the reader is referred to two previous review articles, one by Ferri[2] and the other by Waltrup et al.,[3] and to papers by Waltrup and Billig[4] and Northam and Anderson.[5] Several papers containing general discussions of the evolution and status of CFD techniques have been written recently.[6-10] Of particular note is the paper by Kutler,[11] which presents a very broad and well-informed perspective of current and future capabilities and from which much of the information in the following discussion was obtained.

As previously mentioned, the major pacing items in the development of CFD have been the rapidly increasing capabilities of the digital computer and the development of increasingly sophisticated algorithms for the solution of the fluid dynamic equations of motion. The development of computational capabilities for fluid dynamics had its origins in the mid-1950s with the availability of the first digital computer, the IBM 704, and with the introduction of Lax' first-order method[12] for time-accurate finite-volume or finite-difference solutions of hyperbolic differential equations. Lax' solution method permitted reasonably accurate solutions of flowfields that contained shocks. As a point of reference, it was the late 1950s when the feasibility of supersonic combustion for ramjet engines was analytically demonstrated to be an attractive propulsion concept for flight above Mach 5.[13-15] There was also interest in using supersonic combustion to augment the lift of supersonic airfoils.[16,17]

The development of CFD capabilities gained momentum in the early and mid-1960s, during which time there were two order-of-magnitude increases in computer speed with the introduction of the IBM 7090 class computer in 1960 and then the CDC 6600 in 1964. In addition, a second-order accurate solution algorithm published by Lax and Wendroff,[18] when applied to the fluid dynamic governing equations, resulted in much better resolution of entire flowfields and, in particular, flow regions in the vicinity of shocks. In 1966, Morretti and Abbett[19] published results from the application of a time-dependent computational solution to the previously unsolved supersonic blunt body problem. The time asymptotic solution of the unsteady governing equations for the mixed subsonic-supersonic blunt body flowfield proved to be an important development in the treatment of flowfields in which the equations governing the steady flow are of a mixed type (i.e., elliptic and hyperbolic) or are completely elliptic in space as are the Reynolds-averaged Navier-Stokes equations.

The mid-1960s was a time of heightened activity in the development and testing of scramjet engine concepts in the United States, with the major sponsors being the Navy, the Air Force, and NASA.[3] From 1965 to 1968, a major effort evolved at the NASA Langley Research Center, resulting in the Hypersonic Research Engine (HRE) program, which had as its objective the demonstration and advancement of scramjet technology as a means of propulsion for manned vehicles. The Air Force sponsored three major scramjet development programs, including the General Applied Science Laboratory low-speed, fixed-geometry scramjet[20] program and the Marquardt Company dual-mode scramjet program,[21] both of which took place from 1964 to 1968, and a development and test effort focused on a water-cooled variable-geometry scramjet at the United Aircraft Research Laboratory[22] from 1965 to 1968. Navy work in scramjets intensified in the mid-1960s with an increased effort in the Supersonic Combustion Ramjet Air-breathing Missile (SCRAM) program under the direction of The Johns Hopkins University Applied Physics Laboratory (JHU/APL).

From the late 1960s to the mid-1970s, there were several very important strides in the development of CFD as a practical engineering tool in fluid dynamics. Perhaps the most notable algorithm introduced was by MacCormack in his classic 1969 paper on the effect of viscosity on hypervelocity impact cratering.[23] The MacCormack explicit solution technique, which is a variation on the Lax-Wendroff second-order accurate scheme, uses a predictor-corrector approach to achieve second-order accuracy while using only first-order accurate finite differences. Since it is easily applied to complex flowfields and is very robust, the MacCormack scheme quickly gained widespread acceptance throughout the CFD community and is still widely used today. In fact, many of the codes currently being applied to scramjet analysis, which will be discussed in subsequent sections of this paper, use the MacCormack scheme as the basic solution algorithm. Implicit solution techniques appeared in the 1970s and were motivated by the increasing interest in solving the Reynolds-averaged Navier-Stokes equations, which require fine-grid resolution in the near-wall regions to resolve the boundary layer. The implicit schemes are not subject to severe stability restrictions on the size of the marching step imposed on explicit schemes by the compressed grid and, therefore, may achieve a convergence rate increase of one to two orders of magnitude. However, the implicit solution procedure is generally more difficult to program and requires more computation time per step. The pioneering work on implicit schemes for finite-difference solutions to the fluid dynamic equations of motion was performed by Briley and McDonald[24] and Beam and Warming.[25] The order-of-magnitude increases in computer speed continued with the introduction of the CDC 7600 in 1968, but more important to the dramatic increase in the capability of CFD was the introduction of the vector processor in the mid-1970s, which made the solution of multidimensional viscous flows much more practical.

During the 1970s, scramjet research went through a period of adjustment. The development of the HRE by NASA was halted upon termination of the X-15 program in 1968, and NASA's interest then focused on the development of an airframe-integrated hydrogen-fueled modular scramjet concept for hypersonic propulsion.[26,27] The Navy SCRAM program continued at JHU/APL until 1977, when concern over shipboard storage of highly reactive and toxic fuels dictated a change in concept. The result was the development of the integral rocket dual combustor ramjet concept, which permitted use of conventional hydrocarbon fuels by incorporating a fuel-rich gas generator to preburn the fuel for a main supersonic combustor.[28] The Air Force program in scramjets was basically nonexistent throughout the 1970s.

It was during the 1970s that techniques in computational fluid dynamics were considered for use in the analysis of scramjet components. A considerable amount of pioneering work on the solution of the combustor and nozzle flowfields for hydrogen-fueled supersonic combustion ramjet engines utilizing multistep/multicomponent chemical kinetics models was done under the direction of Ferri.[29] A viscous characteristics approach was used that split the solution into strongly coupled parabolic mixing and hyperbolic wave-solver solutions. Upgrades to the solution technique were made by Spalding and his colleagues[30] to increase the efficiency of the viscous characteristics approach by applying a fully implicit methodology. However, the original solution approach required lengthy iterations to arrive at a local pressure field, and it represented waves in a highly smeared manner.[31] A very significant effort in the application of finite-difference solutions of the viscous flow in a high-speed inlet was the work of Knight.[32] Knight utilized the explicit MacCormack predictor-corrector algorithm to perform a time-dependent solution of the two-dimensional, full, mean, compressible Navier-Stokes equations in conservation form. Computed flowfield properties compared favorably with experimental measurements for two different supersonic inlet designs, and the solution predicted the qualitative behavior of shock-wave/boundary-layer interactions. Although highly successful in proving the feasibility of the computational approach, this capability was by no means a practical engineering tool, given the computer technology of the time. In fact, Knight stated that the two flowfield solutions obtained required 21 and 33.8 h of CPU time on a CDC 6600.

The application of techniques in computational fluid dynamics to the analysis of air-breathing engine components has undergone a maturation process that started in the late 1970s and is continuing today. In 1980, Knight published an improved numerical algorithm for high-speed inlets[33] that still used MacCormack's explicit scheme but included a modified treatment of the viscous sublayer and transition wall region for turbulent boundary layers and incorporated time-split finite-difference operators to obtain an order of magnitude increase in efficiency over his previous code. Using a hybrid explicit-implicit numerical algorithm,[34] in 1983, Knight developed a vectorized code for the solution of the three-dimensional Navier-Stokes equations in high-speed inlets[35] and presented a short course on its use.[36]

The capability for viscous analysis of high-speed inlets using a parabolized Navier-Stokes code called PEPSIS (Partially Elliptic Streamwise Implicit Supersonic) has been developed under the direction of the NASA Lewis Research Center. The original code development was performed by Buggeln et al.[37] The parabolized Navier-Stokes (PNS) equations assume that the streamwise viscous terms are negligible which, along with the special treatment of the subsonic region of the boundary layer, removes the spatial ellipticity from the steady form of the equations and permits a solution by streamwise marching computational techniques. The major advantage of the PEPSIS code, and the PNS codes in general, is that they solve a steady form of the governing equations and are much more efficient than time-dependent codes for solution of the unsteady Navier-Stokes equations. However, large regions of subsonic or separated flow cannot be resolved since upstream influences become important. The PEPSIS code has been used to analyze the supersonic flow (i.e., upstream of the terminal shock region) through high-speed inlets typical of jet fighters[38-39] and to analyze the axisymmetric Lewis 40-60 inlet.[40] In 1983, Anderson presented a design methodology for the solution of the flowfield in a complete supersonic inlet, using a zonal approach and applying the PEPSIS code to the region upstream of the terminal shock, an unsteady Navier-Stokes code (MINT) in the terminal shock region, and a globally iterated PNS code (PEPSIG) for the subsonic diffuser flow.[41]

Although the capabilities developed with the Knight code and the PEPSIS code are applicable to scramjet inlet analysis, they were not used as actual analysis tools for scramjet applications until recently. The work by Knight has, thus far, basically been a research effort in the development of a computational capability, and the PEPSIS code has been mainly used for low supersonic Mach number, high Reynolds number flows through aircraft inlets. These codes are currently being investigated for applications related to scramjet engines operating at high Mach number and low Reynolds number flight conditions.

Work in the application of modern CFD techniques to the design and analysis of actual scramjet engines has been under way since the late 1970s at the NASA Langley Research Center and The Johns Hopkins University Applied Physics Laboratory. The Langley efforts have focused on the use of finite-difference codes for the solution of flows applicable to their hydrogen-fueled modular scramjet engine. The inlet analysis capability, initially published in the early 1980s by Kumar,[42] involved the development of a two-dimensional, time-dependent Navier-Stokes code, NASCRIN. Subsequent code developments included a three-dimensional inviscid code[43] and finally a three-dimensional Navier-Stokes code, SCRAMIN.[44] Work by Drummond et al.[45] to develop a finite-difference capability for the analysis of the Langley modular scramjet combustor initially involved the development of a code for the time-dependent solution of the two-dimensional Navier-Stokes and species equations about a slotted perpendicular hydrogen fuel injector to study the fluid dynamic effects of perpendicular hydrogen injection into a supersonic airstream.[46] The results of a parametric investigation of staged fuel injection were published in 1981.[47] Subsequent development included the incorporation of a complete reaction hydrogen chemistry model and application of the resulting capability, a code called TWODLE, to the analysis of geometries typical of the Langley engine.[48-49] The inlet and combustor capabilities have been combined to perform a viscous two-dimensional solution of the flow through a complete modular scramjet engine configuration.[50] The most recent work has involved the incorporation of an equilibrium air capability for the inlet codes and the extension of the combustor code to include an 18-equation H_2-air chemistry model in a version of the code called SPARK.

Work at JHU/APL in the application of CFD techniques for scramjet applications has focused on the development of practical design and analysis tools for the development of the dual combustion ramjet (DCR) engine concept. Early work by Griffin led to the development and application of efficient spatial marching inviscid finite-difference codes for the three-dimensional analysis of various scramjet inlet configurations. An application of the developed capability to the parametric investigation of the air capture, additive drag, and cowl drag of three candidate scramjet inlet configurations was presented in 1981.[51] Similar codes for internal scramjet inlet flowfields played an instrumental part in the development of the multiple inward-turning scoop inlet concept for the DCR engine. Recent improvements by Van Wie have included the incorporation of equilibrium air chemistry and integration of the CFD codes with automated numerical optimization techniques for the design of optimal inlet and nozzle geometries.[52] Although inviscid codes are very useful as a design tool, any quantitative inlet analysis must include the effects of viscosity; therefore, a NASA Ames three-dimensional PNS code[53] was modified for internal axisymmetric geometries[54] and applied to the design and analysis of DCR inlet concepts.[55] A cooperative effort between JHU/APL and NASA Langley, established in 1983, has permitted application of the time-dependent Navier-Stokes codes developed by Kumar to the analysis of the DCR inlet.[56] The analysis of a supersonic DCR combustor using a zonal solution methodology for the application of finite-difference computational techniques to the hydrocarbon-air combustion process was presented by Schetz et al. in 1981.[57] The same basic computational approach has also been extended to the analysis of scramjet nozzles.[58] The inviscid and PNS inlet analysis capabilities, as well as the combustor analysis capabilities, have been used to provide performance estimates for the combined DCR engine cycle.[59] A concept for the application of these capabilities to the design of a complete tactical missile has been presented.[60]

Considerable expertise for the application of space-marching viscous codes has been recently developed at SAIC. Two-dimensional planar or axisymmetric inlet flowfields can be solved using the sister PNS codes SCRAMP and SCRINT for external and internal flows, respectively.[61] Solutions can be generated assuming either perfect gas or equilibrium air. Combustor and nozzle flowfields are solved using the SCORCH and the SCHNOZ PNS codes,[62] respectively, both of which include the effects of H_2-air chemical kinetics. By integrating the aforementioned capabilities, solutions have been generated for a complete, generic scramjet engine flowfield at high hypersonic Mach numbers.

A major factor that has permitted the solution of the multidimensional Navier-Stokes equations has been the growing availability of powerful vector-processing supercomputers with steadily increasing speed and storage capacity. The Cray X-MP, which became available in 1983, and the Cyber 205 provide performance capabilities several orders of magnitude greater than their counterpart scalar machines. A significant improvement in future computer capabilities related to CFD applications is the development of the numerical aerodynamic simulator (NAS), which is a supercomputer complex being developed at NASA Ames Research Center. The goal of the NAS, dedicated in March 1987, is to provide the computational capability to simulate the realistic flow about a complete aircraft. The capability of the NAS relative to past and present computers is presented in Fig. 1.[11] An excellent discussion of the planned NAS computational facility is presented by Bailey and Ballhaus.[63]

The growth of CFD since its embryonic stages in the late 1950s and, in particular, in the last 15 years, has been truly remarkable. Mature technologies are now becoming available to solve the flowfield through a complete scramjet engine. The methodology for, and some examples of, the application of CFD techniques to the design and analysis of scramjet inlets, combustors, and nozzles at NASA Langley and JHU/APL will be discussed in detail in subsequent sections of this article.

CFD Applied to Scramjets

Flowfield Modeling

The flowfield through a scramjet engine can be described by the Reynolds-averaged, unsteady Navier-Stokes (UNS) equations.[44] In addition to these governing fluid dynamic equations, relations between thermodynamic properties are required. To obtain closure, an approximation for the laminar viscosity coefficient is obtained often using Sutherland's law, and an estimate of the turbulent viscosity coefficient, using a model appropriate for the particular flowfield of interest, is also required. For analysis of the combustor and nozzle flowfield, solution of the species continuity equations is required to account properly for the transport of species produced by chemical reactions. It also becomes necessary to incorporate a model for either hydrogen or hydrocarbon combustion. The turbulence models, as well as the combustion models, will be discussed in subsequent sections; so, for now, let us concentrate on the solution to the fluid dynamic governing equations.

Application of the Reynolds-averaged Navier-Stokes equations to a scramjet flowfield requires transformation to a body-fitted coordinate system as discussed in Ref. 44. Although the flowfields of interest are usually steady, the Navier-Stokes equations are elliptic in space (i.e., disturbances at a point can be felt throughout), which precludes numerical solution by straightforward spatially marched finite-difference schemes. The generally accepted method of solution uses a time-dependent approach in which the unsteady equations (which are temporally hyperbolic), along with the appropriate auxiliary equations necessary to obtain closure, are iterated in time until a steady-state solution is obtained. Convergence to a steady state often requires several thousand iterations which, coupled with the enormous number of grid points required to properly resolve a three-dimensional viscous flow, results in computer run times of several hours, even on the latest supercomputers. Therefore, in the application of CFD techniques to the design and analysis of scramjet engines, various levels of approximation to the Navier-Stokes equations are used to obtain relatively efficient flowfield solutions for different phases of the design process (Fig. 2).

Parabolized Navier-Stokes (PNS)

As just mentioned, the Navier-Stokes equations are elliptic in space; for supersonic flow, however, the equations can be made parabolic in the streamwise direction by assuming that the streamwise viscous terms are negligible and by special treatment of the pressure in the subsonic, near-wall region of the boundary layer. The steady equations can then be solved using a streamwise marching finite-difference scheme, resulting in a dramatic increase in efficiency over a time-dependent solution. It is emphasized that most PNS solutions require that a supersonic core flow exist, and they become unstable if a large region of subsonic flow or any streamwise separation occurs.

Reduction in Dimension

Often, characteristics of the flowfield of interest are such that no simplifying fluid dynamic assumptions can be made. To make a solution more economic and still be able to predict the proper fluid dynamic flow characteristics, it may be desirable to make the geometric assumption of two-dimensional planar or axisymmetric flow. This is especially true in a combustor where, along with the fluid dynamics, combustion models requiring significant computational overhead must be solved.

Euler/Boundary-Layer Superposition

High Reynolds number supersonic flows may often be characterized as mainly inviscid with a thin viscous boundary layer near a solid boundary. Solution of the inviscid (Euler) equations results in a great saving in computer time, not only

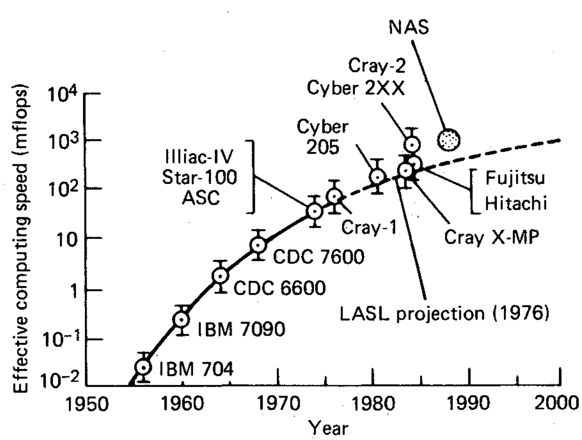


Fig. 1 Trend of effective speed of general-purpose mainframe computers.

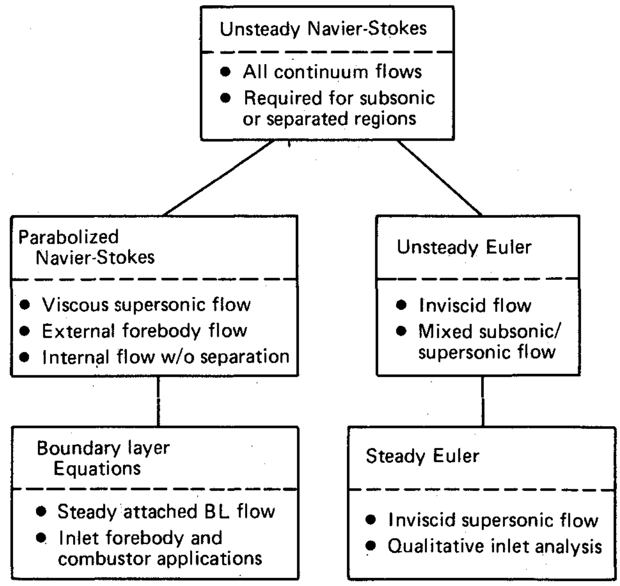


Fig. 2 Hierarchy of fluid dynamic governing equations.

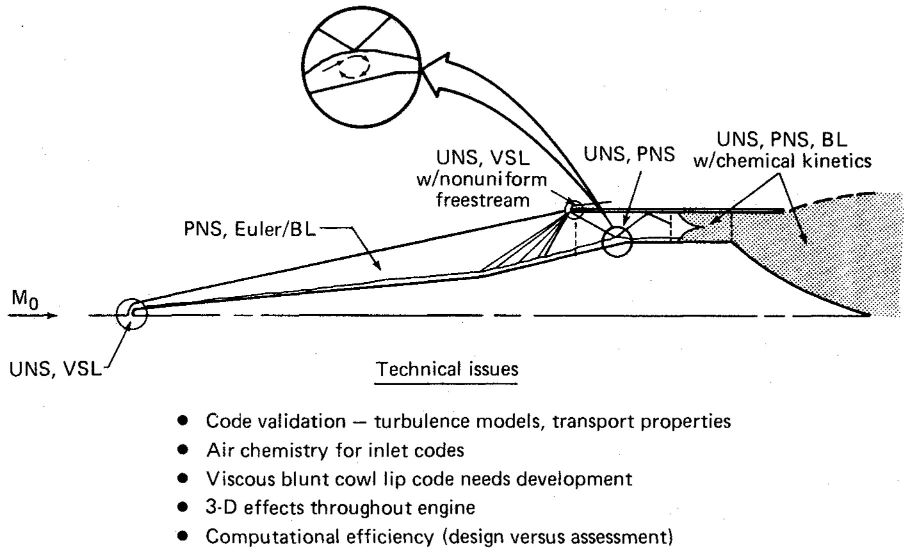


Fig. 3 Schematic of a suggested zonal methodology for scramjet flowfield soltuions.

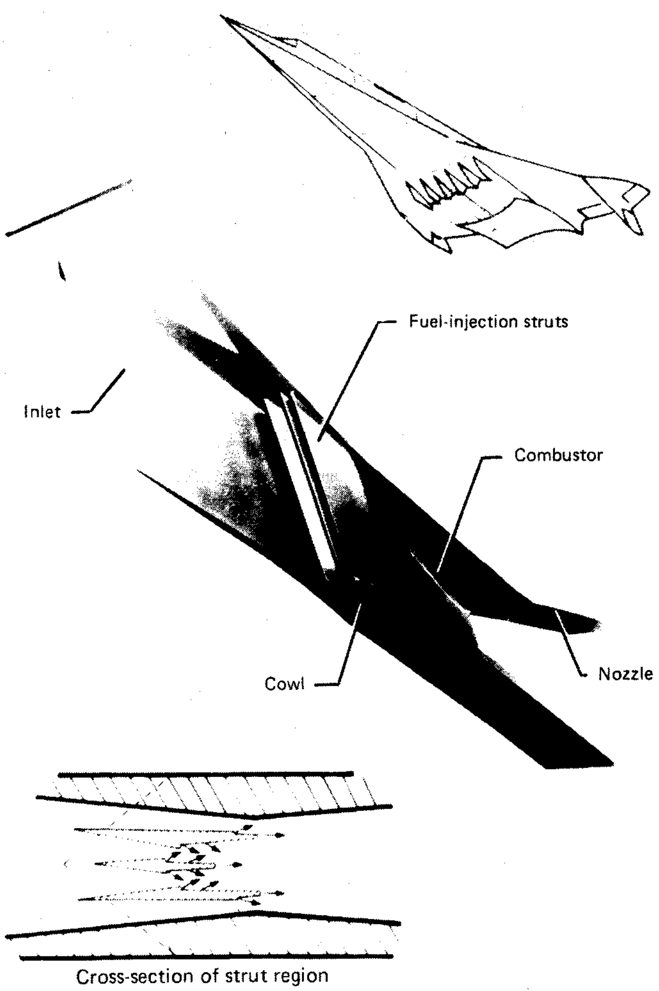


Fig. 4 NASA/Langley scramjet engine module.

because of the reduction in the number of computations but also because of the dramatic reduction in the number of grid points necessary to resolve the flowfield properly. If, in addition, the flowfield is supersonic throughout, the solution can be obtained by a spatially marched finite-difference scheme requiring only a few seconds of computer time. The resulting inviscid solutions can be coupled with a boundary-layer code to assess viscous effects; however, it is noted that for hypersonic flows, iteration is usually required to obtain accurate results using Euler/boundary-layer superposition. Regardless, this capability can be quite beneficial in scramjet inlet design, as will be discussed shortly.

Solution Techniques

The explicit predictor-corrector finite-difference algorithm of MacCormack[23] has been used for all the scramjet codes in routine use at NASA Langley and JHU/APL, with the exception of the codes that solve the boundary-layer equations for the combustor and the PNS equations for the inlet. The solution algorithm is used for both inviscid supersonic steady marching codes and for the time-dependent solution of the Navier-Stokes equations. For inviscid applications, the technique is both easy to program and efficient. For time-dependent viscous calculations, it has gained renewed popularity with the development of vector-processing supercomputers. In addition, the inlet codes of Kumar also have the option of using MacCormack's implicit scheme[64] for increased efficiency.

When applied to a time-dependent solution of the governing equations, the MacCormack algorithm computes the flowfield at time $(t+dt)$ from a known flowfield at time (t) such that the entire flowfield is marched in time to a converged steady-state solution. A similar approach is used for the steady supersonic marching applications, except the solution in plane $(x+dx)$ is calculated using the solutions in plane (x) with the marching direction being the streamwise spatial coordinate instead of time. These solution techniques require the addition of a fourth-order numerical damping term to smooth oscillations in the neighborhood of strong shocks.

The PNS solution algorithm uses an implicit noniterative finite-difference procedure formulated by Schiff and Steger[65] and the alternating direction implicit (ADI) algebraic equation solver of Beam and Warming.[25] The original code formulation by Chaussee and Steger[53] has been applied to external inlet flowfield analyses and has also been modified for internal flow and applied to an axisymmetric scramjet inlet.

Solution Methodology

Even with the relatively simple geometries associated with scramjet engines, there exists a complete range of complex fluid dynamic and chemical kinetic phenomena that must be accounted for. A flowfield solution through a complete scramjet engine can be made practical through the implementation of a zonal solution methodology shown schematically in Fig. 3. The implementation of this solution methodology would involve application of parabolized Navier-Stokes equations in regions of attached supersonic flow in the inlet, combustor, and nozzle and resort to use of the unsteady Navier-Stokes (UNS) equations in regions of flow separation or subsonic pockets likely to occur because of strong shock wave/boundary-layer interactions in the inlet and in the vicinity of nonaxial fuel injection in the combustor. Blunt leading edges on the forebody and the cowl lip are also handled with UNS or more efficient viscous shock layer (VSL) codes when applicable. Euler/boundary-layer superposition can be used for parametric design investigations in the inlet, and superposition using PNS or UNS codes with coarse grids and boundary-layer codes employing very fine grids has application in the combustor and nozzle. For hypersonic flight Mach numbers above 7, the inlet codes must account for equilibrium air chemistry. The combustor codes must include, preferably, finite rate fuel/air chemical kinetics; however, global combustion models do have application when the reactions can be assumed to be mixing-controlled and for some parametric design investigations. Figure 3 also lists some of the important technical issues to be addressed for the proper application of CFD codes to scramjet analysis.

Applications

The most effective way to highlight the application of CFD techniques to scramjet engine analysis is to present examples of, and a discussion of, the techniques applied to the analysis of the various engine components. As mentioned earlier, the emphasis is placed on capabilities in use at NASA Langley and JHU/APL and applied to scramjet engines under development at those laboratories.

An engine concept developed at NASA Langley for a hypersonic cruise vehicle is an airframe-integrated, hydrogen-fueled scramjet in which the entire engine system is divided into several identical rectangular modules. An example of one such module is shown with the sideplates removed in Fig. 4. The module has a fixed-geometry inlet with swept wedge-shaped sidewalls which, along with a recess in the cowl, allow starting over a wide range of flight Mach numbers. In the module shown, the compression is completed by three struts, which also provide locations for the injection of gaseous hydrogen fuel. The fuel-air mixture reacts in the combustor portion of the module, and the effluent is expanded through the nozzle. Computational techniques have been used to analyze the entire engine flowfield.

The dual combustor ramjet (DCR) engine under development at JHU/APL for the propulsion of hypersonic missiles is shown schematically in Fig. 5.[28] A small portion of the captured air is mixed with all of the hydrocarbon fuel in a fuel-rich subsonic dump combustor, which preheats the fuel for burning with the remainder of the air in the supersonic combustor. A multiple inward-turning scoop inlet concept in which four scoops provide air to the supersonic combustor and two scoops provide air to the gas generator is currently being investigated for the DCR engine. Techniques in CFD have been used in the design and analysis of the inlet, the supersonic combustor, and the nozzle.

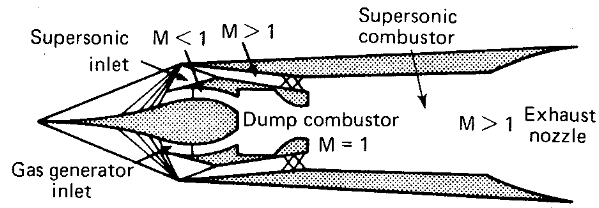


Fig. 5 Schematic illustration of a hypersonic dual combustion ramjet.

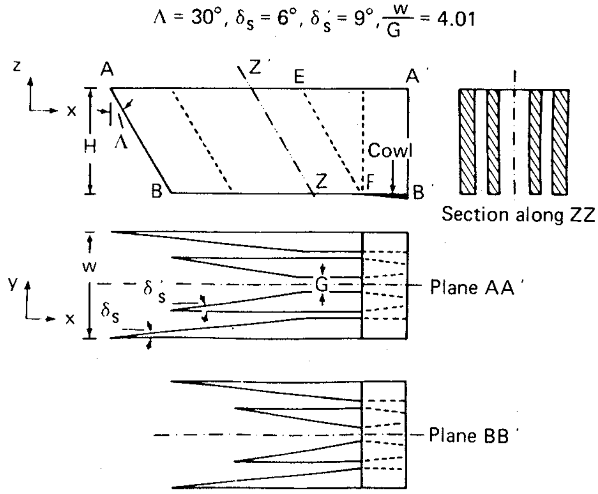


Fig. 6 Line diagram of the NASA two-strut scramjet inlet configuration.

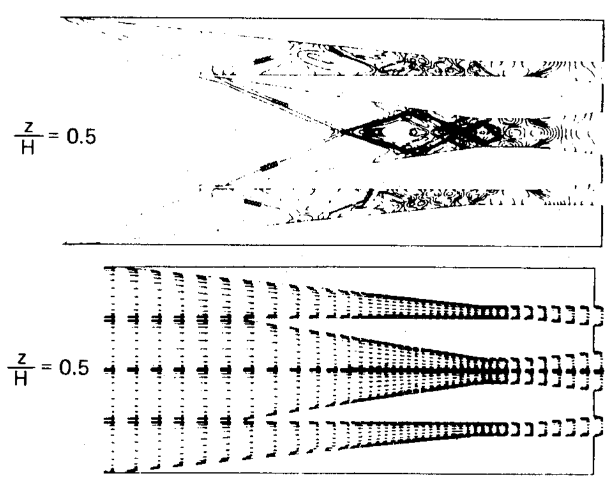


Fig. 7 Pressure contour plot and velocity vector field at mid-inlet height.

Inlet Applications

Two- and three-dimensional computational codes for the time-dependent solution of the unsteady Navier-Stokes (or Euler) equations have been developed by Kumar for application to the analysis of flow through scramjet inlets.[42,44] These codes are operational on the Control Data VPS 32 at NASA Langley and have been optimized to take maximum advantage of the computer's vector-processing capability. Turbulence is modeled using the algebraic two-layer eddy viscosity model of Baldwin and Lomax[66] and, as previously discussed, the MacCormack predictor-corrector solution algorithm is applied.

An application of the three-dimensional code to the analysis of the modular engine is the solution of the flow in a two-strut inlet shown in the line diagram of Fig. 6.[67] Special treatment of the computational boundary upstream of the cowl accounts for the interaction of the internal flow with the external flow. Flowfield solutions were generated using a geometry for which experimental data are available and simulating the experimental conditions of $M_1 = 4.03$, $p_1 = 8724$ N/m^2, and $T_1 = 70$ K. The calculation was initially performed with 80,000 grid points on the VPS 32 and required approximately 100 min of run time to obtain a converged solution. To improve resolution in regions of high gradients near the walls and inlet throat, a solution has been generated using 275,000 grid points, requiring 8 h of run time. The code achieves a compute rate of 0.7×10^{-5} s per grid point per time step on the VPS 32.

Some representative results from the latter calculation are shown in Figs. 7 and 8. The velocity vector field and the static pressure contours in a plane located at mid-inlet height are presented in Fig. 7. Several regions of separation caused by the shock/boundary-layer interaction on the inlet surfaces are evident in the velocity vector diagram. The pressure contour plot clearly shows the complex shock and expansion wave structure in the inlet. The surface pressure distributions on the inlet sidewall and on the inner strut surfaces are shown to agree very well with experiment[68] in Fig. 8. The results of this analysis demonstrate the validity of the application of CFD techniques to the solution of the complete Navier-Stokes equations for a scramjet inlet using state-of-the-art computer technology. Use of a time-dependent code for solution of the complete equations permits resolution of important shock/boundary-layer interactions and the resulting regions of separation.

Although the capability just described provides an extremely valuable analysis resource, the solutions require several hours of run time even on the latest supercomputers. Therefore, the application of computational techniques as a practical engineering tool for the design of scramjet inlets requires a more conservative approach using appropriate approximations to the complete Navier-Stokes equations in the various phases of the design process. The tools currently in use at JHU/APL for the design and analysis of a supersonic inlet for the DCR engine include inviscid codes integrated with numerical optimization procedures for parametric design investigations, PNS codes for preliminary assessment of viscous effects on inlet performance, and Kumar's unsteady Navier-Stokes (UNS) codes for quantitative performance analysis of developed inlet designs.

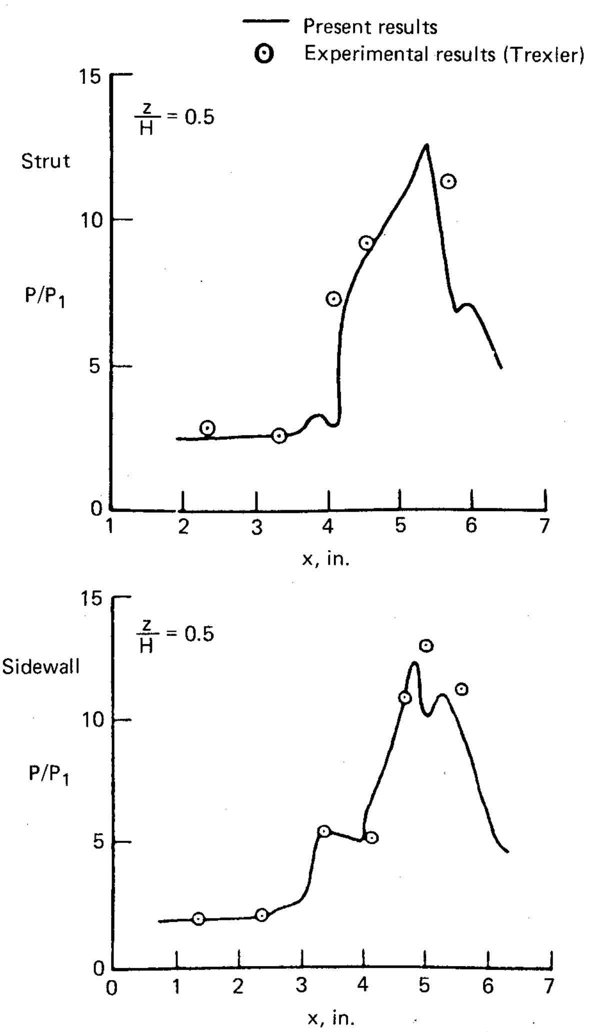


Fig. 8 Pressure distribution on the inlet strut and sidewall surfaces.

Preliminary inlet design investigations for a scramjet engine can entail evaluation of several candidate inlet configurations (chin, annular, etc.) in an effort to determine the one design most applicable to the particular engine under development. On choosing a preferred inlet configuration, it is often desirable to investigate parametrically various design variables (i.e., forebody bluntness, cowl angle, etc.) as to their effect on inlet performance. This phase of the design process, which previously required extensive wind-tunnel testing, can now be accomplished with computational tools. In particular, inexpensive steady-state inviscid marching codes for supersonic flow, such as those described by Kutler,[69] have been routinely used for parametric investigations of inlet performance. The flowfield solutions for a desired inlet can be processed to provide the inlet designer with qualitative estimates of important parameters such as air capture ratio, additive drag, cowl wave drag, and internal kinetic energy efficiency. An example of the application of such a capability is shown in Fig. 9, where the air capture ratio for an axisymmetric supersonic inlet is determined as a function of the nose radius on the sphere-cone forebody. Details of these calculations and more applications of the inviscid codes are presented in Ref. 59.

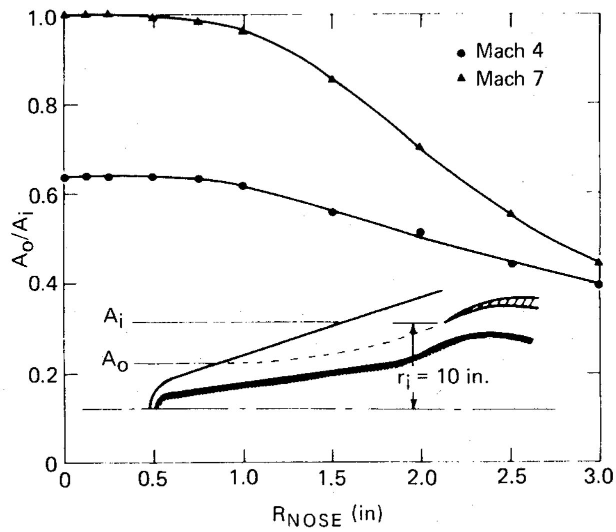


Fig. 9 Air capture vs sphere-cone nose radius for a typical scramjet inlet.

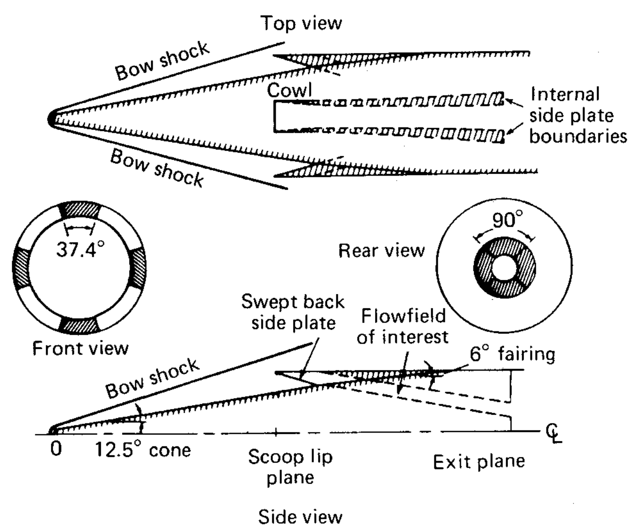


Fig. 10 Schematic illustration of a multiple inward-turning scoop (MITS) scramjet inlet.

As just summarized, inviscid analysis techniques, when used within their realm of applicability, can provide a very valuable screening tool for preliminary inlet design. However, inviscid codes are limited by the very nature of the inviscid approximation and, in most cases, the performance estimates must be considered qualitative. Several important examples in which accounting for viscous effects is essential in scramjet inlet applications include hypersonic flight regimes where viscous-inviscid interactions are important; flight at angle of attack, where boundary-layer effects on the leeward side of the inlet may dominate performance characteristics; and internal flowfields for which boundary-layer effects, including shock/boundary-layer interactions, play a major role in determining the inlet flow structure and the resulting inlet performance.

To address these factors and provide a quantitative analysis for flowfields typical of the multiple, inward-turning scoop (MITS) scramjet inlet shown in Fig. 10, the zonal solution methodology discussed earlier is applied. The blunt-nose region of the forebody is solved using an axisymmetric UNS code, which provides a starting plane for a PNS solution over the conical forebody. Angle-of-attack starting planes are generated by rotating the axisymmetric blunt-body solution. The internal flow in a single scoop has been solved first using the PNS code to obtain preliminary estimates of boundary-layer displacement effects, and then using the UNS code to resolve expected regions of boundary-layer separation due to reflections of the cowl lip shock. Application of this inlet analysis approach has resulted in predictions that agree reasonably well with wind-tunnel measurements and has provided an invaluable diagnostic tool for analysis of the inlet flowfield structure. The internal flowfield structure, as calculated using Kumar's three-dimensional Navier-Stokes code, for an inlet operating at Mach 4, zero angle-of-attack flight conditions is shown in the velocity vector diagram and the pressure contour plot of Fig. 11. The inlet kinetic energy efficiency, plotted as a function of axial station in the inlet for the viscous computation, is compared with the corresponding inviscid prediction in Fig. 12. The importance of viscous effects on inlet performance is clearly evident from the significantly lower inlet efficiency.

Closer inspection of Fig. 12 shows a relatively constant reduction in inlet efficiency due to the effects of viscosity upstream of the inlet throat and in the downstream section of the diffuser; however, in the region of the inlet throat, the losses are much greater than what might be expected due solely to viscous effects. It was hypothesized that the additional loss resulted from the regions of separation, which act as effective bodies and thereby modify the internal shock structure. To determine the validity of this hypothesis, a computational experiment was conducted in which the separated regions were modeled as solid bodies (Fig. 13) and the inviscid solution was repeated. A decrease in inlet efficiency in the throat region of the inlet, similar in trend to that of the viscous calculation, was predicted by the inviscid calculation using the effective inlet contour (Fig. 14). This result seems to validate the original hypothesis as to the source of the additional inlet loss and demonstrates the usefulness of CFD as a diagnostic tool for the analysis of scramjet inlet performance.

As discussed, the use of CFD techniques in scramjet inlet design and analysis provides a valuable tool for all phases of the design process. By no means has the wind tunnel been eliminated as an essential requirement for inlet design, but the maturation of capabilities in CFD has reduced the dependence on testing by nearly eliminating the need for parametric experimental investigations in the preliminary design phase. CFD techniques have also proved to be a valuable diagnostic tool with which to complement and interpret experimental data.

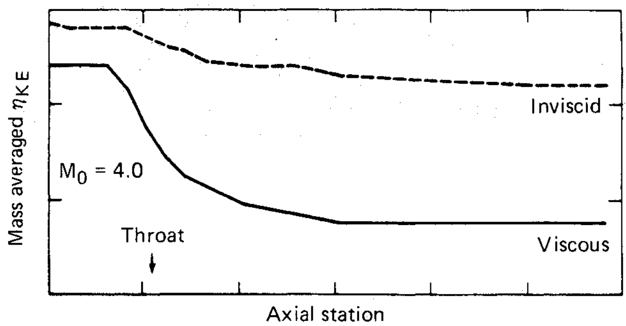


Fig. 12 Axial distribution of kinetic energy efficiency for the MITS inlet.

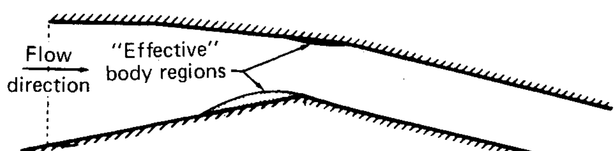


Fig. 13 Effective inlet contour used to evaluate the effect of separation on performance.

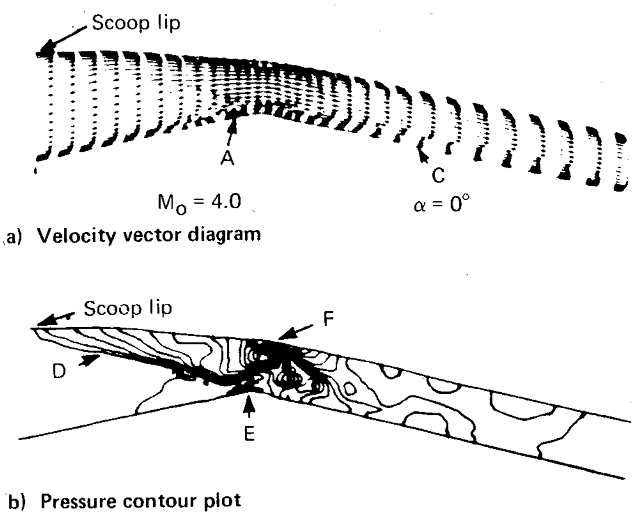


Fig. 11 Velocity vector field and pressure contour plot near the MITS inlet throat.

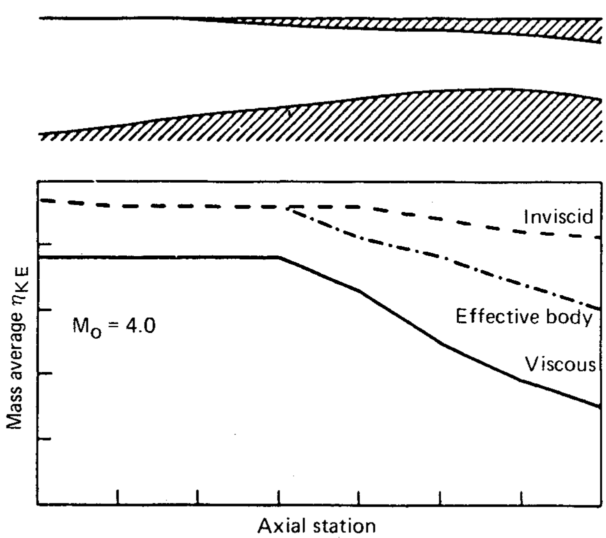


Fig. 14 Axial distribution of kinetic energy efficiency for the effective inlet geometry.

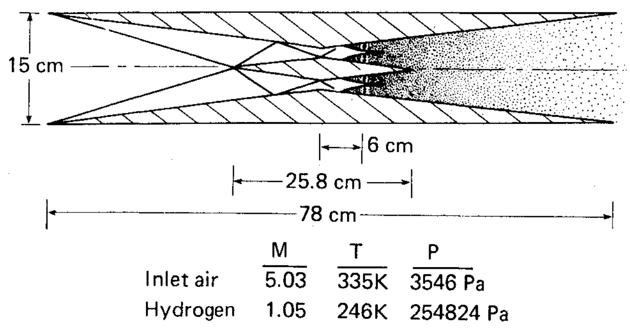


Fig. 15 Schematic of a NASA scramjet engine with expected shock structure and regions of combustion.

Combustor Applications

The flowfield in a scramjet combustor, although predominately supersonic, is often characterized by regions of embedded subsonic and recirculating flow to which the unsteady Navier-Stokes equations must be applied. In addition, the fluid dynamics must be coupled with a combustion model to simulate properly the chemical reactions that occur when the fuel/air mixture is burned. Incorporation of a combustion model with the fluid dynamics results in a significant increase in the amount of computations necessary due to the increase in the number of governing equations. Also, the characteristic time for the chemical reactions is usually much less than that for the fluid dynamics so the size of the time step must be greatly reduced to maintain stability, or the iteration algorithm must be modified to allow for larger time steps. Although some work is in progress for the development of a three-dimensional capability, current computer technology has limited the practical application of CFD capabilities to scramjet combustors to two-dimensional or axisymmetric analyses. With this in mind, computational codes for the analysis of hydrogen/air combustion in the NASA Langley modular scramjet and for the analysis of hydrocarbon/air combustion in the main combustor of the JHU/APL dual combustion ramjet have been developed and are in routine use.

An unsteady Navier-Stokes code, TWODLE, has been developed at NASA Langley for the analysis of the combustor flowfield in the NASA modular scramjet engine. The code solves the two-dimensional form of the Navier-Stokes equations along with one or more additional transport equations describing the species present in the flow. Both the unsplit[23] and the time-split[70] versions of MacCormack's predictor-corrector finite-difference technique have been used for integration of the governing equations. No-slip velocity boundary conditions are used at the walls, which are also assumed to be adiabatic and noncatalytic. The turbulent viscosity is calculated using the algebraic eddy viscosity model of Baldwin and Lomax.[66]

In TWODLE, combustion of the hydrogen/air mixture is modeled using a complete reaction model in which instantaneous reaction is assumed at any point where both fuel and air are present. The extent of the reaction is determined by the amount of fuel present in a fuel-lean condition and by the amount of oxidant present in a fuel-rich condition. No reaction is allowed when the fraction of hydrogen in air is less than 4% by volume. It is pointed out that the complete reaction model is applicable only when the reaction is mixing-controlled rather than controlled by kinetics and implementation of a more general combustor model for finite-rate hydrogen/air chemistry using nine species equations has recently been incorporated in a version of the code called SPARK. Further details of the complete reaction combustor model used for the results discussed herein are presented in Ref. 48.

The code described has been applied to the solution of the flow through a two-dimensional scramjet engine module shown schematically, with expected inlet shock locations and regions of combustion noted, in Fig. 15. This geometry is a two-dimensional projection of an actual scramjet engine design. Flow enters the inlet at $M=5.03$, $T=335$ K, and $p=3546$ Pa. Hydrogen is injected transversely to the main flow from the single fuel injection strut and from the engine sidewalls at $M=1.05$, $T=246$ K, and $p=254{,}824$ Pa. This injection results in a fuel equivalence ratio of 1. The four transverse injectors (Fig. 15) are located 6 cm downstream of the engine cross-sectional minimum and are each 0.1 cm wide.

Results for an engine solution using a grid with 87 nodes in the streamwise direction and 30 nodes in the transverse direction are summarized in Figs. 16–18. Figure 16 shows the magnified velocity vector field in the engine. The air entering the inlet is turned by shocks emanating from the inlet leading edges. (Shocks are indicated by a coalescence of pressure contour lines in Fig. 17.) These shocks strike near the strut leading edge and coalesce with shocks produced by the strut. The resulting shocks then have sufficient strength to separate the boundary layer when they reach the engine sidewalls, as indicated by a reversal of the velocity vectors; induced shocks then result from the separation regions.

The transverse hydrogen fuel injectors located downstream of the minimum can also be seen in the figures, along with their associated flow separations leading and trailing the injectors. The flow becomes subsonic near the fuel injectors due to both the main flow blockage by the injectors and the heat release from chemical reaction. Some reaction takes place in the separated regions ahead of the injectors; significant reaction occurs downstream of the injectors.

Computed contours of combustion-related flowfield properties are presented in Figs. 18a–c. The effect of chemical reaction on the static temperature field can be seen in Fig. 18a. Contours are plotted for temperatures of 400, 600, 1000, 1400, 1800, 2200, and 2600 K. Maximum temperatures are reached in the separated regions just ahead of the injectors and at the edge of the hydrogen/air mixing layer just downstream of the injectors. It is at these locations that the highest degree of reaction occurs. Figure 18b describes the total hydrogen mass fraction distribution, including gaseous hydrogen and the hydrogen in water, in the combustor. Contours are plotted for 0.1, 1, 10, 20, and 40% by mass of hydrogen. This distribution provides a good indication of fuel/air mixing. The highest concentrations of hydrogen lie closest to the strut and engine sidewalls, but concentrations of around 1% hydrogen do penetrate across the flow, indicating reasonable mixing but at a considerable distance downstream of the injectors. The water mass fraction distribution in the combustor for contours of 0.1, 1, 10, 20, and 24% is shown in Fig. 18c. The results agree with the previous figures in that the maximum concentration of water lies midway through the fuel/air mixing layers where the degree of reaction is the highest and the temperatures peak. However, values of water concentration midway across cross-sectional planes in the combustor are still quite low.

For the conditions shown here, it is evident that the injectors are not of sufficient strength to penetrate across the flow at the point of injection, and the opposing mixing layers do not meet until approximately 11 cm downstream of the injectors. Reaction occurs across the entire flowfield beyond this point, although the highest degree of reaction still occurs in the interior of the mixing layers. The results indicate a need to relocate or reconfigure the injectors to improve fuel penetration and mixing. Ideally, the fuel should be well mixed with air throughout the flowfield downstream of the injectors. The program allows the degree of mixing to be evaluated over a range of configurations so that the injector design and location can be optimized. This capability is demonstrated in the following example.

In an attempt to improve fuel/air mixing, the injectors were relocated 4.7 cm upstream of the previous injector location. Results for this case are given in Fig. 19, which is a

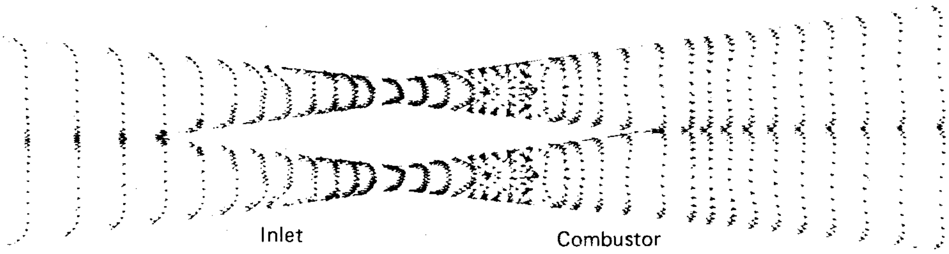
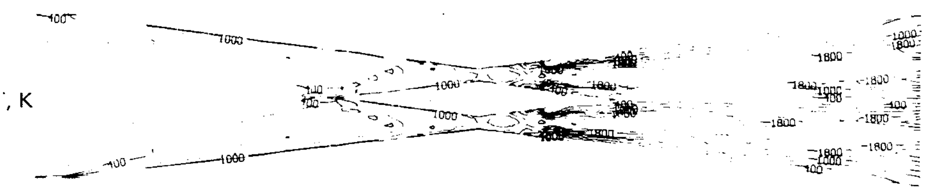
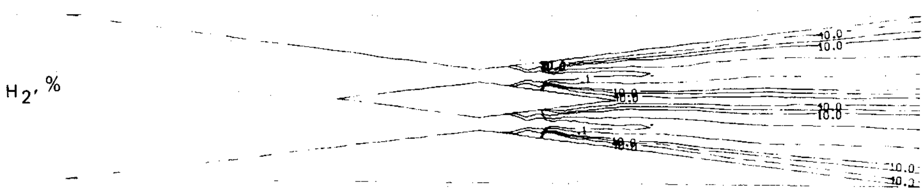


Fig. 16 Magnified velocity vector field in the NASA engine.

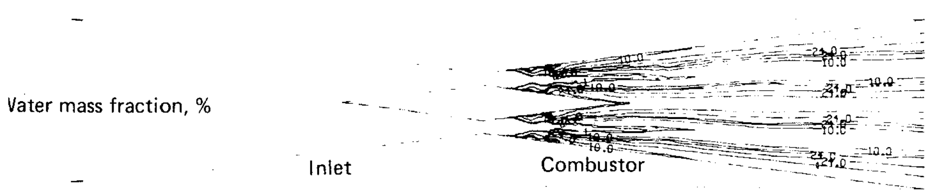
Fig. 17 Computed static pressure contours in the NASA engine.



a) Computed static temperature contours



b) Computed total hydrogen mass fraction contours



c) Computed water mass fraction contours

Fig. 18 Combustion-related flowfield contours computed for the NASA engine.

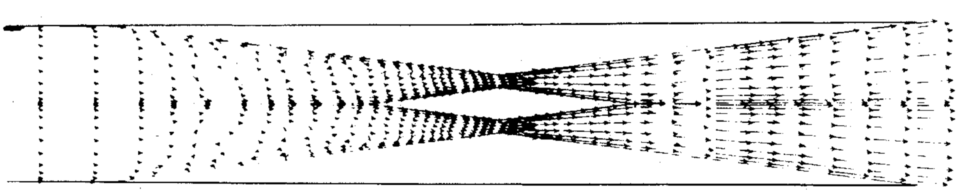


Fig. 19 Computed velocity vector field in the NASA engine with choked flow.

velocity vector field plot of the engine flowfield. Note the large regions of separated flow that persist in the inlet nearly to the leading edge. The separation results from choking at the engine minimum, which results in a strong adverse pressure gradient along the engine sidewalls. The fuel injector location for this engine geometry would not be acceptable. The calculation was terminated once the massive separations and a severe mass imbalance formed. The program could be used to find the resulting flowfield after the engine choked by redefining the physical domain and boundary conditions. Such information is not that helpful, however, once it has been determined that a particular engine configuration is not useful. To investigate the source of the choking problem, the aforementioned calculation was repeated without chemical reaction. This calculation proceeded normally with mass conserved and without the appearance of any significant separation. Therefore, it was concluded that thermal choking was produced by chemical reaction and subsequent heat release near the injectors.

The preceding two-dimensional engine calculations demonstrate a capability for the analytical prediction of scramjet performance over a range of conditions typical of hypersonic propulsion applications. The computer program was used to calculate successfully details of the turbulent reacting flowfield in a scramjet and has provided information needed to actually design an engine. Further extension of the program to three dimensions, along with the addition of improved chemistry and turbulence models, is necessary before actual three-dimensional engine configurations can be considered. The successful analysis of a two-dimensional scramjet engine model problem indicated that such an extension was feasible, and work is currently under way to develop and extend the program. Again, computer growth will probably be the pacing item for a complete engine analysis.

A schematic illustration of the model used to analyze the flowfield in the JHU/APL dual combustor ramjet (DCR) is shown in Fig. 20. Afterburning of the fuel-rich discharge from the gas generator occurs in the main supersonic combustor, generating a precombustion shock train in the supersonic flow supplied by the inlet. Separated zones are present on both the combustor outer wall and in the base of the gas generator nozzle. To render this complex flow tractable, a composite model has been adopted using a zonal solution methodology. The central core flow is solved by applying a mixing and burning code, which is based on the parabolic boundary-layer equations, using a relatively coarse grid (25–30 points) in the transverse direction. Slip boundary conditions are imposed at the wall. The flowfield solution is obtained for an axisymmetric combustor by marching downstream from the initial data plane using an explicit finite-difference scheme. Solution of the entire combustor requires approximately 1000 axial steps. Initial conditions and an axial pressure distribution are provided from integral analysis techniques[71] based on estimates of overall heat release, and the solution is iterated if the calculated overall heat release is significantly different from the estimate. Any *a priori* knowledge of the effective combustor area distribution is not required. In fact, $A(x)$ is an output of the code, and its proximity to the actual combustor geometry being investigated provides a measure of the validity of the computed flowfield solution.

The resulting solution of the core flow is evaluated at the wall to obtain edge conditions, including the turbulent kinetic energy (TKE), for a wall boundary-layer analysis that is then used to calculate wall skin friction and heat transfer. For this analysis, the boundary-layer equations are solved using an implicit, axial marching, finite-difference algorithm with 101 grid points from the wall to the edge of the boundary layer. The code used is a greatly extended version of the boundary-layer code presented in Ref. 72. The modified code includes multicomponent diffusion of species and calculates the correct laminar mixture properties in the near-wall region of the boundary layer.

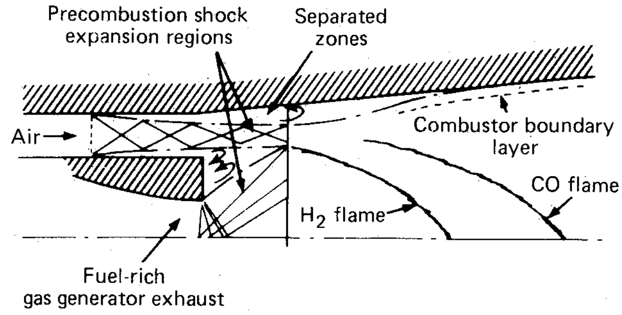


Fig. 20 Schematic illustration of the flow model used in the composite DCR engine analysis.

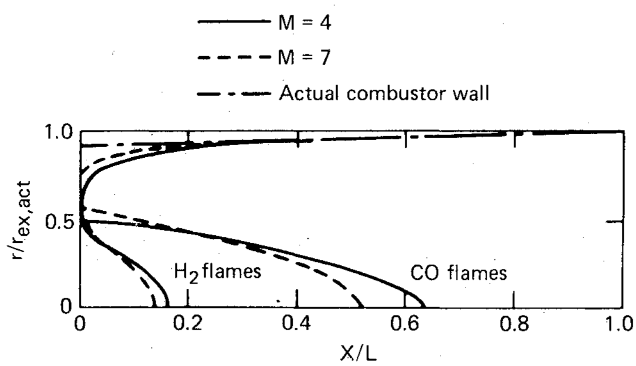


Fig. 21 Computed duct contours and flame shapes for the DCR engine at Mach 4 and 7 with $ER = 0.5$.

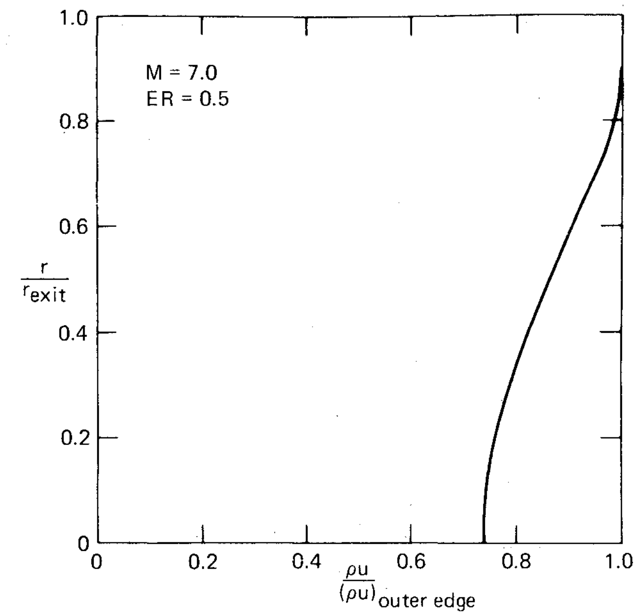


Fig. 22 Computed exit plane mass flux profile for the DCR engine at Mach 7 with $ER = 0.5$.

Combustion was modeled in both the mixing and burning code for the core flow and the boundary-layer code for the near-wall flow using equilibrium chemical reactions including H,C,O, and N atoms and the molecular combinations of O_2, N_2, H_2, CO, CO_2, H_2O, OH, and NO. This system is reduced for improved computational efficiency by assuming a double flame sheet model,[73] which considers an inner H_2 flame and an outer CO flame with only the O_2, H_2, N_2, H_2O, CO, and CO_2 molecules present. The flame sheet model has been shown to provide accurate results below a temperature of 2500 K. If the computed temperature at particular grid points is above 2500 K, then a complete equilibrium calculation is made before proceeding to the next axial step.

Turbulent transport properties are calculated for the core flow using a Prandtl energy method[74] for jet mixing that is based on an eddy viscosity model tied directly to the turbulent kinetic energy throughout the flow. In the boundary-layer analysis at the wall, a turbulent transport model is used that applies a conservation equation for the TKE suitable for the wall region. Details of the turbulence model, as well as additional details for the combustion models, can be found in Refs. 57 and 59.

The composite combustor analysis capability was applied to a representative DCR engine (recall Fig. 5) at simulated flight conditions of Mach 7 and Mach 4 using Shelldyne H fuel at an overall equivalence ratio (ER) of 0.5. Calculations were made starting with the application of the integral analysis technique to determine an axial pressure distribution $P(x)$ and the initial conditions for a specified combustor wall shape. With this information, the mixing and burning code was then run. The calculated shape of the flow boundary for the given $P(x)$ is shown in Fig. 21, along with the shape of the H_2 and CO flames for the two operating conditions. The size of the flames is evident from the figure, indicating that the combustor length is sufficient to provide for complete combustion. The validity of the modeling approach using zonal finite-difference solutions based on a pressure distribution calculated by an integral analysis is demonstrated by the close proximity of the computational boundary to the actual combustor geometry. The qualitative features of the separated zone on the outer boundary of the shock interaction region at the combustor entrance are also evident.

The exit plane mass flux profile results shown in Fig. 22 indicate that even though combustion has been completed, residence time in the combustor is not adequate to yield uniform property profiles in the combustor exit plane. The degree of thrust decrement due to nonuniformity depends on the exhaust nozzle geometry and can be determined using the techniques discussed in the subsequent section of this paper. Some detailed profiles across the duct at an axial station near the end of the H_2 flame are shown for the Mach 7 case in Fig. 23. The full equilibrium calculations were made at several points to resolve properly behavior that could not be predicted with the flame sheet model. The combustor core flow calculation was evaluated at the boundary to provide input edge conditions of velocity, temperature, molecular mass fraction of each species, and turbulent kinetic energy, for the subsequent boundary-layer calculation.

Boundary-layer calculations were run for the Mach 7 case to an axial distance about halfway down the combustor. The wall temperature was assumed to be constant at 2000°R. The variation of the skin friction coefficient based on local edge conditions, C_{fe}, along the combustor wall is shown in Fig. 24. The region labeled compression ahead of the combustor results from the simplified representation of the interaction-shock region that was used to provide initial conditions for the calculation. The skin friction values in the combustor are rather high for this high Reynolds number flow, and they do not decay roughly at $x^{-1/5}$ as would be the case for a constant-pressure flow. Thus, the indirect effects of the flame on the boundary layer are seen to be significant. The dimensional wall heat-transfer rate also shown in Fig. 24 is more sensitive to the local pressure gradient. These heat-transfer rates are rather large, even though this case is lean overall and the flame does not impinge on the wall.

The modular analysis approach just described for the DCR engine provides the capability to investigate combustor performance for a wide variety of conditions. Important properties can be calculated, including combustor exit flow profiles, flame structure, and wall skin friction and heat transfer.

Nozzle Applications

The zonal solution methodology applied to the DCR supersonic combustor has been applied in the development of a computational capability for analysis of scramjet nozzles. The simplifying assumption that the radial pressure gradient is negligible, made in the combustor core flow to permit use of the parabolic boundary-layer equations, cannot be made for the nozzle. As a result, the core flow of the nozzle is calculated using the axisymmetric form of the unsteady Navier-Stokes equations. The equations are integrated in time using MacCormack's predictor-corrector technique with a grid of 20–25 radial points and 30–40 axial

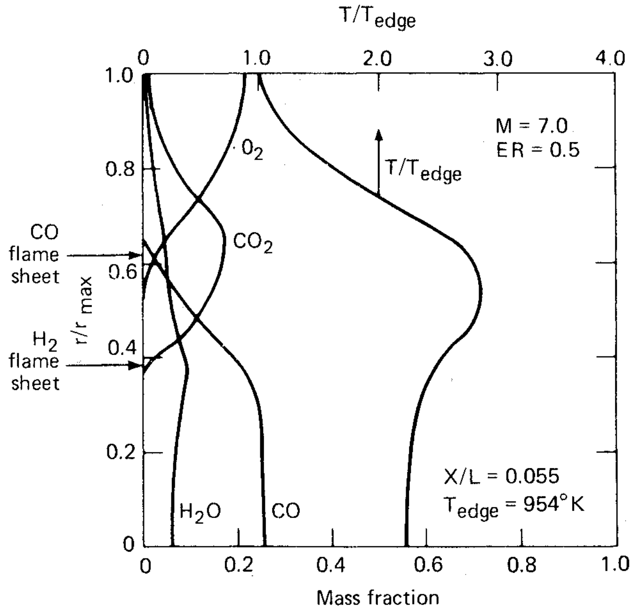


Fig. 23 Computed flow profiles near the end of the H_2 flame for the DCR engine at Mach 7 with $ER = 0.5$.

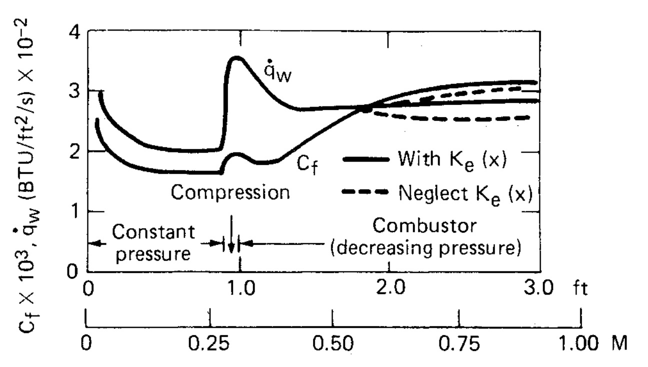


Fig. 24 Computed skin-friction coefficient and heat-transfer rate for the DCR engine at Mach 7 with $ER = 0.5$.

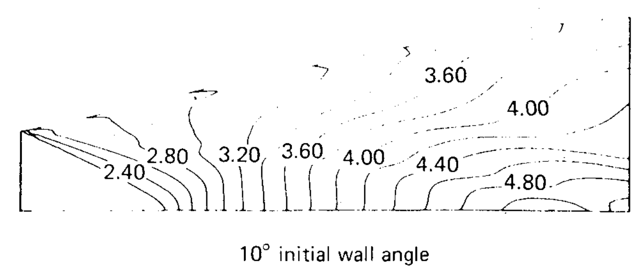


Fig.25 Preliminary DCR nozzle contour with inviscid Mach number profiles calculated for an "equivalent" uniform inflow.

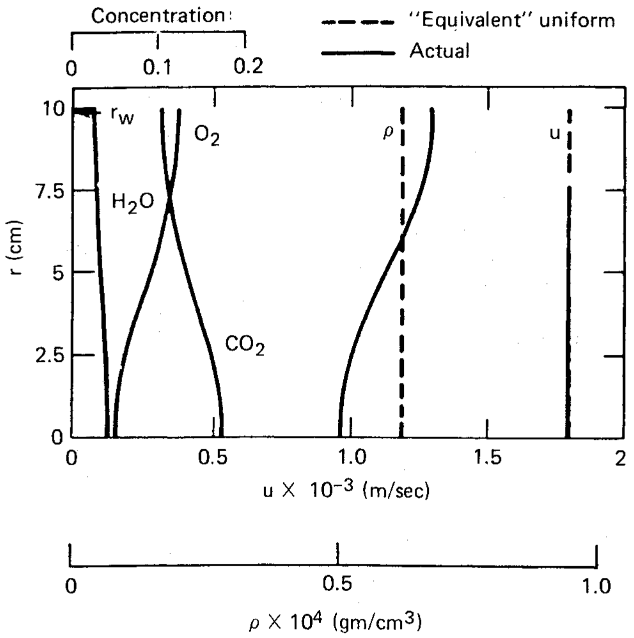


Fig. 26 Combustor exit profiles used as inflow for nozzle flowfield calculations.

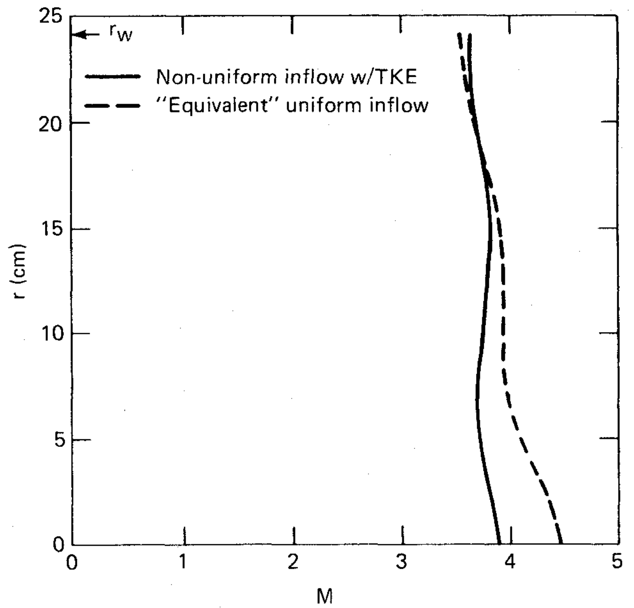


Fig. 27 Calculated exit plane Mach number profiles for uniform and nonuniform inflow.

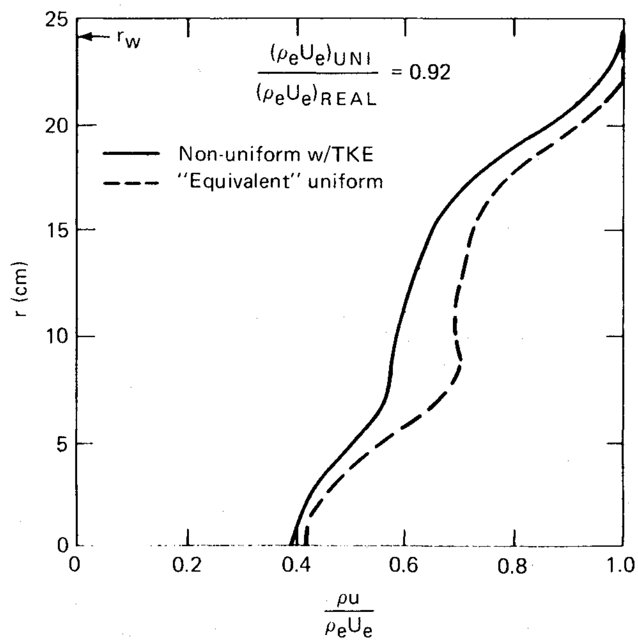


Fig. 28 Calculated exit plane mass flow profiles for uniform and nonuniform flow.

points. The wall boundary conditions allow for velocity slip, and the wall properties are evaluated and used as input to the same boundary-layer code presented for the combustor analysis. The code used for the core flow calculation is a modified form of the VNAP2 code developed by Cline at Los Alamos.[75] Chemical reactions were modeled using the same species assumed present in the combustor (see earlier discussion). Assumptions of frozen flow, local equilibrium, or simple finite-rate chemistry are made when appropriate. The finite-rate scheme uses a flame sheet model for production of water from hydrogen and oxygen molecules, and a rate equation for the production of carbon dioxide from carbon monoxide and oxygen molecules. Again, the turbulent transport properties are calculated using a partial differential equation for the TKE distribution.

The developed computational capability for nozzle analysis has been applied to candidate DCR nozzle configurations. One such nozzle is shown in Fig. 25 with Mach contours from an inviscid calculation, integrated with a numerical optimization on nozzle thrust coefficient, performed to determine the design contour. The main objective in developing the CFD capability is to evaluate the effect of nozzle inflow nonuniformities and the turbulent mixing of those nonuniformities on nozzle performance. A flowfield solution for the nozzle shown in Fig. 25 was run using both an "equivalent" uniform inflow and the actual nonuniform combustor exit flow shown in Fig. 26. The effect of nozzle inflow nonuniformity on Mach number is shown in Fig. 27, indicating that the nonuniform inflow actually produces a more uniform Mach number profile at the exit. However, the exit mass flow distribution (Fig. 28) for the nonuniform inflow is more nonuniform than that for the uniform inflow case. This information can be used, along with similar information for stream thrust, to evaluate the performance of various nozzle designs. More details and additional application of the nozzle analysis capability are presented in Ref. 58.

Future Directions

Computational techniques for scramjet inlet analysis are the most mature of the engine component analysis tools. Codes are already in use that solve the complete unsteady Reynolds-averaged Navier-Stokes equations, with turbulence, for which no simplifying fluid dynamic assumptions are made. The current efforts are focused on the application of this capability to a broader and more geometrically complex range of problems and on the development and application of new, more efficient numerical solution techniques. A discussion of the status and directions of some of these new numerical techniques has been presented by MacCormack.[76] The goal of these techniques is to greatly increase the solution convergence rate, and an example of a viscous supersonic blunt-body flowfield solution is given in which the number of iterations for convergence is reduced by an order of magnitude when compared to a conventional implicit solution. A promising algorithm development is also emerging in the application of upwind relaxation schemes, which recognize the hyperbolic nature of the inviscid form of the governing equations, to the solution of the Navier-Stokes equations. The work of Thomas and Walters[77] uses an upwind scheme for the thin-layer Navier-Stokes equations and applies a line Gauss-Siedel relaxation approach. Future efforts will focus on applying these upwind schemes to the solution of internal scramjet flowfields. An alternative approach to the solution of the unsteady Navier-Stokes equations for flows with zones of subsonic or recirculating flow is to solve the PNS equations using a global iteration scheme.[78] Efforts to develop a globally iterated PNS capability for the accurate solution of scramjet inlet flowfields that include large regions of subsonic or separated flow are currently under way. These efforts are discussed in Refs. 79 and 80.

Analysis of scramjet combustor flowfields has, thus far, been limited to fairly simple chemistry and turbulence models to make the problems tractable on available computer systems. Chemistry is often modeled with either complete reaction (mixing-controlled) or global finite-rate chemistry models, which assume mixing-controlled combustion, and turbulence modeled with either algebraic eddy viscosity, turbulent kinetic energy, or k-ϵ models. However, chemical reaction throughout a scramjet combustor is controlled, in general, by kinetics and not mixing. The recent development of the SPARK code with its 18-equation finite-rate chemistry model for H_2-air combustion and more thorough treatment of the turbulent dissipation terms takes a step in the direction of applying more elaborate models for unsteady Navier-Stokes applications.

Research to develop improved reacting flow models is proceeding along three directions, including the development of detailed finite-rate kinetic models, the extension of advanced turbulence modeling techniques used for nonreacting flows to include reaction, and the development of improved numerical algorithms for solving the resulting equations that govern turbulent reacting flows. Chemistry is being modeled with up to a 9-species, 18-equation finite-rate model to describe the hydrogen/air reactions of importance in current scramjet designs. To analyze the flow in the upstream portion of the combustor, particularly near flameholders, it is necessary to consider important ignition species to ensure that they are continuously produced in sufficient quantity to allow reaction to proceed to later species. A detailed diffusive fuel/air mixing model has also been developed to describe small-scale mixing of the various chemical species. This model considers the multicomponent diffusion of each species into all other species on diffusive scales.

Phenomena present in chemically reacting flows place additional complications upon the already difficult problem of modeling turbulence in nonreacting flows. Direct simulation of turbulent reacting flows would provide many of the details needed to understand important phenomena, but computer limitations restrict even the computation of nonreacting flows in the turbulent regime. Chemical reactions introduce additional important scales that further complicate the problem; however, large eddy simulation relieves some of the scaling constraints, since only the large-scale phenomena would be computed. Small-scale phenomena are described in this scheme using subgrid scale models developed for nonreacting flow, and work has begun to extend some of these models to reacting flow.[81] To perform needed engineering calculations for the design of scramjet combustors, however, phenomenological models of turbulence must still be employed. Direct simulation will not be extended into a realistic Reynolds number regime for some time to come, and the large eddy simulation is not amenable to the routine day-to-day calculations necessary for design of a scramjet combustor. Rather, it appears that the most reasonable approach is to use the more detailed techniques to understand important phenomena present in turbulent chemically reacting flow. The resulting numerical data can then be used to improve the phenomenological models used in design calculations.

Reacting flow experiments carried out to provide data for improving both finite-rate kinetics models and turbulence models are discussed in detail in Ref. 82. The information available from experiment has improved markedly in recent years due to advancements in nonintrusive diagnostics. The new diagnostic techniques, including laser Doppler velocimetry (LDV) and coherent anti-Stokes ramon spectroscopy (CARS), allow measurement of flow properties without any flow perturbations, and they yield data on fluctuation properties as well as on mean quantities. This experimental data, used in conjunction with the numerical data discussed in the previous paragraph, should be extremely useful in improving and validating models used to compute turbulent reacting flows.

Work is also currently under way to develop improved numerical algorithms for solving the equations governing chemically reacting flows. This system of equations is characterized by a widely separated range of time scales that causes the system to be numerically stiff. This stiffness comes about from both the time scales associated with the evolution of each chemical species and the diffusive stiffness introduced by the fine spatial resolution required to resolve high spatial gradients present in chemically reacting flows. There have been a number of papers published on the solution of stiff systems of equations describing chemically reacting flows, and the reader is referred to Refs. 83–92 for a more detailed discussion. The resolution of large spatial gradients can be enhanced by implementing methods of higher order than those that have been applied in past work. Spectral methods fall into this class of higher-order methods and are proving to be an effective technique for investigating basic combustion phenomena. With proper application, these high-order approximations can produce extremely accurate numerical solutions, resulting in a significant reduction in the required number of grid points needed relative to a finite-difference computation. Spectral methods still represent a relatively new method for solving chemically reacting flows, and the approach is not without difficulty. Problems still remain when dealing with complex geometries and shocks in supersonic flows. The method does offer much promise for future work, however, particularly with regard to achieving a better understanding of basic combustion phenomena. The reader is referred to Refs. 93–95 for a more detailed discussion on the application of spectral methods to chemically reacting flows and to Refs. 96–98 for a further discussion of spectral methods themselves.

Conclusions

The evolution of computational fluid dynamics can be traced back to the late 1950s, about the same time research in scramjet engines for hypersonic propulsion was beginning. The two technologies evolved independently through the next two decades. Maturing CFD technology, brought about by the explosive growth in capability over the past 15 years, coupled with a resurgence in interest in scramjet propulsion in the 1980s, has resulted in the development of valuable computational tools for the design and analysis of scramjet engines. Three-dimensional computational techniques including efficient inviscid supersonic marching codes, parabolized Navier-Stokes codes, and unsteady Navier-Stokes codes are currently in routine use for the design and analysis of scramjet inlets. The capability to compute the two-dimensional planar or axisymmetric flow in a supersonic combustor using finite-difference codes has been developed for both hydrogen- and hydrocarbon-fueled engines. A very significant capability for the application of CFD techniques to scramjet engine design and analysis has been developed and has proved to be of great value in the design and analysis of scramjet engines at the NASA Langley Research Center and at the Johns Hopkins University Applied Physics Laboratory.

Recently, interest in scramjet propulsion has reached a level reminiscent of the 1960s, and the prospect for large-scale research and development efforts in future years appears to be very promising. Unlike the earlier efforts, however, CFD is playing an instrumental role in scramjet development programs. As a result of this increased interest, the future evolution of CFD capabilities for scramjet applications will continue to accelerate greatly with major efforts throughout government, industry, and university laboratories. Prominent areas of development will be in the application of new numerical techniques for solution of the governing equations, more sophisticated models for turbulence and combustion, and the development of an integrated capability for the solution of the flow through a complete scramjet engine. It is emphasized that the application of CFD techniques remains an art as well as a science and that meaningful results depend not only on code integrity but also on user experience. It is important that the user be knowledgable about the physics being modeled, as well as the code being applied. With such knowledge the application of CFD techniques to scramjet engine analysis can be an invaluable engineering tool.

References

[1]Anderson, J.D., "A Survey of Modern Research in Hypersonic Aerodynamics," AIAA Paper 84-1578, June 1984.

[2]Ferri, A., "Review of Scramjet Propulsion Technology," *Journal of Aircraft,* Vol. 5, Jan.-Feb. 1968, pp. 3-10.

[3]Waltrup, P.J., Anderson, G.Y., and Stull, F.D., "Supersonic Combustion Ramjet (Scramjet) Development in the United States," 3rd International Symposium on Air Breathing Engines, Munich, FRG, March 1976.

[4]Waltrup, P.J. and Billig, F.S., "Liquid Fueled Supersonic Combustion Ramjets: A Research Perspective of the Past, Present, and Future," AIAA Paper 86-0158, Jan. 1986.

[5]Northam, G.B. and Anderson, G.Y., "Review of NASA/Langley Basic Research on Supersonic Combustion," AIAA Paper 86-0159, Jan. 1986.

[6]Paynter, G.C., "CFD Technology for Installation Design: Forecast for the 80's."

[7]Hankey, W.J., "Research in Computational Aerodynamics," Tech. Rept. AFFDL-TR-3127, Feb. 1980.

[8]Chapman, D.R., "Dryden Lectureship in Research-Computational Aerodynamics Development and Outlook," AIAA Paper 79-0129, Jan. 1979.

[9]Chapman, D.R., "Trends and Pacing Items in Computational Aerodynamics," *Lecture Notes in Physics,* Vol. 141, edited by W.C. Reynolds and R.W. MacCormack, Springer-Verlag, New York, June 1980, pp. 1-12.

[10]Taylor, T.D., "Modern Computational Fluid Dynamics," AIAA Paper 80-0860, May 1980.

[11]Kutler, P., "A Perspective of Theoretical and Applied Computational Fluid Dynamics," *AIAA Journal,* Vol. 23, March 1985, pp. 328-341.

[12]Lax, P.D., "Weak Solutions of Nonlinear Hyperbolic Equations and Their Numerical Computation," *Communications on Pure and Applied Mathematics,* Vol. 7, Feb. 1954, pp. 159-193.

[13]Ferri, A., "Possible Directions of Future Research in Air Breathing Engines," 4th AGARD Colloquium, Milan, Italy, April 4-8, 1960; also, *Combustion and Propulsion—High Mach Number Air Breathing Engines,* edited by A.L. Joumotte, A.M. Rothrock, and A.H. Lefebvre, Pergamon Press, New York.

[14]Dugger, G.L., "Comparison of Hypersonic Ramjet Engines with Subsonic and Supersonic Combustion," 4th AGARD Colloquim, Milan, Italy, April 4-8, 1960.

[15]Dugger, G.L., Deklau, B., Billig, F.S., and Matthews, S., "Summary Report on External Ramjet Program," TG-419, JHU/APL, Laurel, MD, Oct. 1961.

[16]"The Performance of Supersonic Burning Engines at Various Equivalence Ratios and Its Use as a Powerplant for Auxiliary Jet Lift," Marquardt Corp., Rept. MR-20,006, Nov. 1958.

[17]Mager, A., "Supersonic Airfoil Performance with Small Heat Addition," *Journal of the Aerospace Sciences,* Vol. 26, Feb. 1959.

[18]Lax, P.D. and Wendroff, B., "Difference Schemes for Hyperbolic Equations with High Order Accuracy," *Communications on Pure and Applied Mathematics,* Vol. 17, Aug. 1964, pp. 381-398.

[19]Moretti, G. and Abbett, M., "A Time Dependent Computational Method for Blunt Body Flows," *AIAA Journal,* Vol. 4, Dec. 1966, pp. 2136-2141.

[20]"Investigation of the Low Speed Fixed Geometry Supersonic Combustion Ramjet," General Applied Science Laboratories, AFAPL-TR-66-139, March 1967.

[21]Burnette, T.D., "Dual Mode Scramjet," Marquardt Corp., AFAPL-TR-67-132, June 1968.

[22]Kepler, C.E. and Karanian, A.J., "Hydrogen-Fueled Variable Geometry Scramjet," United Aircraft Research Laboratories, AFAPL-TR-67-150, Jan. 1968.

[23]MacCormack, R.W., "The Effect of Viscosity on Hypervelocity Impact Cratering," AIAA Paper 69-354, April 1969.

[24]Briley, W.R. and McDonald, H., "Solution to the Multi-Dimensional Compressible Navier-Stokes Equations by a Generalized Implicit Method," *Journal of Computational Physics,* Vol. 24, Aug. 1977, pp. 372-397.

[25]Beam, R. and Warming, R.F., "An Implicit Factored Scheme for the Compressible Navier-Stokes Equations," *AIAA Journal,* Vol. 16, April 1978, pp. 393-402.

[26]Beach, H.L. Jr., "Hypersonic Propulsion," Paper XII, NASA CP-2092, May 1979.

[27]Hunt, J.L., Lawing, P.L., and Cubbage, J.M., "Conceptual Study of Hypersonic Air-Breathing Missiles," AIAA Paper 78-6, Jan. 1978.

[28]Billig, F.S., Waltrup, P.J., and Stockbridge, R.D., "Integral-Rocket Dual-Combustion Ramjets: A New Propulsion Concept," *Journal of Spacecraft and Rockets,* Vol. 17, Sept.-Oct. 1980.

[29]Ferri, A., "Mixing Controlled Supersonic Combustion," *Annual Review of Fluid Mechanics,* Vol. 5, 1973.

[30]Elghobashi, S.E. and Spalding, D.B., "Equilibrium Chemical Reaction of Supersonic Hydrogen-Air Jets," NASA CR-2725, 1977.

[31]Dash, S.M., private communications, Nov. 25, 1985.

[32]Knight, D.D., "Numerical Simulation of Realistic High-Speed Inlets Using the Navier-Stokes Equations," *AIAA Journal,* Vol. 15, Nov. 1977.

[33]Knight, D.D., "Improved Numerical Simulation of High-Speed Inlets Using the Navier-Stokes Equations," AIAA Paper 80-0383, Jan. 1980.

[34]Knight, D.D., "A Hybrid Explicit-Implicit Numerical Algorithm for the Three-Dimensional Compressible Navier-Stokes Equations," AIAA Paper 83-0223, Jan. 1983.

[35]Knight, D.D., "Calculation of a Simulated 3-D High-Speed Inlet Using the Navier-Stokes Equations," AIAA Paper 83-1165, June 1983.

[36]Knight, D.D., "AFWAL/NASA Short Course on 3-D Navier-Stokes Supersonic Inlet Code," sponsored by AFWAL and NASA/LaRC, Hampton, VA, Jan. 16-18, 1984.

[37]Buggeln, R.C., McDonald, H., and Levy, R., "Development of a Three-Dimensional Supersonic Inlet Flow Analysis," NASA CR 3218, Jan. 1980.

[38]Anderson, B.H. and Towne, C.E., "Numerical Simulation of Supersonic Inlets Using a Three-Dimensional Viscous Flow Analysis," AIAA Paper 80-0384, Jan. 1980.

[39]Benson, T.J., "Application of Three-Dimensional Viscous Marching Analysis to Supersonic Inlets," Advanced Computational Methods Workshop, NASA Lewis, Nov. 1980.

[40]Benson, T.J. and Anderson, B.H., "Validation of a Three-Dimensional Viscous Analysis of Axisymmetric Supersonic Inlet Flowfields," AIAA Paper 83-0135, Jan. 1983.

[41]Anderson, B.H., "Three-Dimensional Design Methodology for Advanced Technology Aircraft Supersonic Inlet Systems," NASA TM 83558, AIAA 22nd Aerospace Sciences Meeting, Reno, NV, Jan. 9-12, 1984.

[42]Kumar, A., "Numerical Analysis of the Scramjet Inlet Flowfield Using Two-Dimensional Navier-Stokes Equations," AIAA Paper 81-185, 1981.

[43]Kumar, A., "Three Dimensional Inviscid Analysis of the Scramjet Inlet Flowfield," AIAA Paper 82-0060, Jan. 1982.

[44]Kumar, A., "Numerical Analysis of a Scramjet Inlet Flowfield Using the Three-Dimensional Navier-Stokes Equations," presented at the 1983 JANNAF Propulsion Meeting, Monterey, CA, Feb. 14-18, 1983.

[45]Drummond, J.P., Rogers, R.C., and Evans, J.S., "Combustor Modelling for Scramjet Engines," the AGARD 54th (B) Specialists' Meeting on Combustor Modelling—Propulsion and Energetics Panel, Cologne, Germany, Oct. 3-5, 1979.

[46]Drummond, J.P., "Numerical Investigation of the Perpendicular Injector Flowfield in a Hydrogen Fueled Scramjet," AIAA Paper 79-1482, July 1979.

[47]Weidner, E.H. and Drummond, J.P., "Numerical Study of Staged Fuel Injection for Supersonic Combustion," *AIAA Journal,* Vol. 20, Oct. 1982, pp. 1426–1431.

[48]Drummond, J.P. and Weidner, E.H., "Numerical Study of a Scramjet Engine Flowfield," *AIAA Journal,* Vol. 20, Sept. 1982, pp. 1182–1187.

[49]Drummond, J.P. and Weidner, E.H., "A Numerical Study of Candidate Transverse Fuel Injector Configurations in the Langley Scramjet Engine," the 17th JANNAF Combustion Meeting, Hampton, VA, Sept. 22-26, 1980.

[50]Kumar, A. and Drummond, J.P., "Numerical Study of Scramjet and Ramjet Flow Fields," Presented at the 1983 JANNAF Propulsion Meeting, Monterey, CA, Feb. 14–18, 1983.

[51]Waltrup, P.J., Billig, F.S., and Stockbridge, R.D., "Engine Sizing and Integration Requirements for Hypersonic Air Breathing Missile Applications," PEP-58-8, AGARD Propulsion and Energetics Panel, 58th Symposium, Ramjets and Ramrockets for Military Applications, London, England, Oct. 26–30, 1981.

[52]Van Wie, D.M., "An Application of Computational Fluid Dynamics to the Design of Optimum Ramjet Powered Missile Components," Ph.D. Dissertation, University of Maryland, University Microfilms Inc., Ann Arbor, MI, May 1986.

[53]Chaussee, D.S. and Steger, J.L., "Three-Dimensional Viscous Flow Field Program, Part 2: Parabolized Navier Stokes Program," FSI Rept. 80-06, Dec. 1980.

[54]White, M.E. and Waltrup, P.J., "Computational Fluid Dynamics for Internal Flows," APL Independent Research and Development, 1983–1985.

[55]Griffin, M.D., "Computational Fluid Dynamics—A Tool for Missile Design," *JHU/APL Technical Digest,* Vol. 4, July–Sept. 1983, pp. 187–195.

[56]White, M.E. and Hagan, J.C., "Computational Viscous Analysis of Supersonic Inlets for the Hypersonic Dual Combustion Ramjet," Report to the Navy Aeroballistics Committee Gas Dynamics Panel, White Oak, MD, Feb. 1984.

[57]Schetz, J.A., Billig, F.S., and Favin, S., "Flowfield Analysis of a Scramjet Combustor with a Coaxial Fuel Jet," *AIAA Journal,* Vol. 20, Sept. 1982, pp. 1268–1274.

[58]Schetz, J.A., Billig, F.S., and Favin, S., "Numerical Solutions of Ramjet Nozzle Flows," AIAA Paper 85-1270, July 1985.

[59]Griffin, M.D., Billig, F.S., and White, M.E., "Applications of Computational Techniques in the Design of Ramjet Engines," presented at the 6th International Symposium on Air Breathing Engines, Paris, France, June 6–10, 1983.

[60]Billig, F.S., White, M.E., and Van Wie, D.M., "Application of CAE and CFD Design Techniques to a Complete Tactical Missile Design," AIAA Paper 84-0387, Jan. 1984.

[61]Krawczeck, W.J., Rajendran, N., and Harris, T.B., "Computational Models for the Analysis and Design of Hypersonic Scramjet Components, Part II: Inlet and Ramp/Forebody Models," AIAA Paper 86-1596, June 1986.

[62]Dash, S.M., Sinha, N., Wolf, D.E., and York, B.J., "Computational Models for the Analysis and Design of Hypersonic Scramjet Components, Part I: Combustor and Nozzle Models," AIAA Paper 86-1595, June 1986.

[63]Bailey, F.R. and Ballhaus, W.F. Jr., "A Large-Scale Computer Facility for Computational Aerodynamics," *IEEE Transactions of Nuclear Science,* Vol. NS-32, Feb. 1985, pp. 202–208.

[64]MacCormack, R.W., "A Numerical Method for Solving the Equations of Compressible Viscous Flow," AIAA Paper 81-0110, Jan. 1981.

[65]Schiff, L.B. and Steger, J.L., "Numerical Simulation of Steady Supersonic Viscous Flow," AIAA Paper 79-0130, Jan. 1979.

[66]Baldwin, B.S. and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, Jan. 1978.

[67]Kumar, A., "Numerical Simulation of Flow Through Scramjet Inlets Using a Three-Dimensional Navier-Stokes Code," AIAA 85-1664, July 1985.

[68]Trexler, C.A., "The Design and Performance at Mach 4.0 of Two-Strut Hypersonic Inlets for an Integrated Scramjet Concept," Proposed NASA TM.

[69]Kutler, P., "Computation of Three-Dimensional Inviscid Supersonic Flows," *Lecture Notes in Physics,* Vol. 41, *Progress in Numerical Fluid Dynamics,* Springer-Verlag, New York, 1975, pp. 287–374.

[70]MacCormack, R.W. and Baldwin, B.S., "A Numerical Method for Solving the Navier-Stokes Equations with Application to Shock-Boundary Layer Interactions," AIAA Paper 75-1, Jan. 1975.

[71]Billig, F.S. and Dugger, G.L., "The Interaction of Shock Waves and Heat Addition in the Design of Supersonic Combustors," 12th International Symposium on Combustion, The Combustion Institute, Pittsburgh, PA, 1969, pp. 1125–1134.

[72]Anderson, E.C. and Lewis, C.H., "Laminar or Turbulent Boundary-Layer Flows of Perfect Gases or Reacting Gas Mixtures in Chemical Equilibrium," NASA CR-1893, Oct. 1971.

[73]Schetz, J.A., "A Simplified Model for the Combustion or Multicomponent Fuels in Air," *Combustion and Flame,* Vol. 15, Aug. 1970, pp. 57–60.

[74]Prandtl, L., "Uber eine neues Formelsystem Für die ausgebildete Turbulenz," Nachrichten der Akademie Wissenschaften, Gottingen, Mathphys, Van Den Loeck & Ruprecht, 1945, pp. 6–19.

[75]Cline, M.C., "VNAP2: A Computer Program for Computation of Two-Dimensional, Time-Dependent, Compressible Turbulent Flow," Los Alamos National Laboratory, LA-8872, Aug. 1981.

[76]MacCormack, R.W., "Current Status of Numerical Solutions of the Navier-Stokes Equations," AIAA Paper 85-0032, Jan. 1984.

[77]Thomas, J.L. and Walters, R.W., "Upwind Relaxation Algorithms for the Navier-Stokes Equations," AIAA Paper 85-1501, July 1985.

[78]Thompson, D.S. and Anderson, D.A., "A Pseudo-Unsteady Approach for Predicting Steady Supersonic Flows," AIAA Paper 87-0541, Jan. 1987.

[79]"The Development of a 3D Viscous Flow Analysis for the Evaluation of Hypersonic Inlets," Defense SBIR Phase II Proposal, AMTEC Engineering Inc., 1985.

[80]Chitsomboon, T., Kumar, A., and Tiwari, S.N., "A Parabolized Navier-Stokes Algorithm for Separated Supersonic Internal Flows," AIAA Paper 85-1411, July 1985.

[81]Ferziger, J.H., personal communication, Stanford University; also AIAA Paper 84-1578, June 1984.

[82]Northam, G.B. and Anderson, G.Y., "Review of NASA/Langley Basic Research on Supersonic Combustion," AIAA Paper 86-0158, Jan. 1986.

[83]Smoot, L.D., Hecker, W.C., and Williams, G.A., "Prediction of Propagating Methane-Air Flames," *Combustion and Flame,* Vol. 26, 1976.

[84]Stalnaker, J.F., Robinson, M.A., Spradley, L.W., Kurzius, S.C., and Thoenes, J., "Development of the General Interpolants Method for the Cyber 200 Series of Computers," Lockheed-Huntsville Research and Engineering Center, Huntsville, AL, TR D867354, Oct. 1983.

[85]Bussing, T.R.A. and Murman, E.M., "A Finite Volume Method for Calculation of Compressed Chemically Reacting Flows," AIAA Paper 85-0331, Jan. 1985.

[86]Gear, G.W., *Numerical Initial Value Problems in Ordinary Differential Equations,* Prentice Hall, Englewood, NJ, 1971.

[87]Hindmarsh, A.C. and Gear, G.W., "Ordinary Differential Equation Solver," Rept. UCID-30001, Lawrence Livermore Laboratory, Livermore, CA, 1974.

[88]Hindmarsh, A.C. and Byrne, G.D., "Episode: An Effective Package for the Integration of Systems of Ordinary Differential Equations," Rept. UCID-30112, Lawrence Livermore Laboratory, Livermore, CA, 1977.

[89]Linger, W. and Willoughby, R.A., "Efficient Integration Methods for Stiff Systems of Ordinary Differential Equations," *SIAM Journal of Numerical Analysis,* Vol. 7, 1970.

[90]Pratt, D.T., "Exponential-Fitted Methods for Integrating Stiff Systems of Ordinary Differential Equations," CPIA Publ. 401, 1984 JANNAF Propulsion Meeting, Feb. 1984.

[91]Bui, T.D., Oppenheim, A.K., and Pratt, D.T., "Recent Advances in Methods for Numerical Solution of O.D.E. Initial Value Problems," Rept. LBL-16943, Lawrence Berkeley Laboratory, University of California, Berkeley, CA, 1983.

[92]Radhakrishnan, K., "Comparison of Numerical Techniques for Integration of Stiff Ordinary Differential Equations Arising in Combustion Chemistry," NASA TP 2372, Oct. 1984.

[93]Drummond, J.P., Hussaini, M.Y., and Zang, T.A., "Spectral Methods for Modeling Supersonic Chemically Reacting Flow Fields," NASA CR 172578, March 1985; also, AIAA Paper 85-0320, Jan. 1985.

[94]McMurtry, P.A., Jou, W.H., Riley, J.J., and Metcalfe, R.W., "Direct Numerical Simulations of a Reacting Mixing Layer with Chemical Heat Release," AIAA Paper 85-0143, Jan. 1985.

[95]Riley, J.J. and Metcalfe, R.W., "Direct Numerical Simulations of Chemically Reacting Turbulent Mixing Layers," AIAA Paper 85-0321, Jan. 1985.

[96]Hussaini, M.Y., Salas, M.D., and Zang, T.A., "Spectral Methods for Inviscid, Compressible Flows," *Advances in Computational Transonics,* edited by W.G. Habashi, Pineridge Press, Swansea, U.K., 1983.

[97]Gottlieb, D. and Orszag, S.A., "Numerical Analysis of Spectral Methods, Theory and Applications," CBMS-NSF, Regional Conference Series in Applied Mathematics, SIAM, 1977.

[98]Hussaini, M.Y., Kopriva, D.A., Salas, M.D., and Zang, T.A., "Spectral Methods for the Euler Equations," *AIAA Journal,* Vol. 23, Jan. 1985, pp. 64–70.