An Assessment of Numerical Solutions of the Compressible Navier-Stokes Equations

J. S. Shang*

Air Force Wright Aeronautical Laboratories, Wright-Patterson Air Force Base, Ohio

Nomenclature

e	= internal energy $e = e_i + ½\bar{u}\cdot\bar{u}$
e_i	= specific internal energy
$\bar{f}$	= force
F, G, H	= vector fluxes
J	= Jacobian of coordinate transformation
k	= molecular heat conductivity
$\bar{n}$	= outward normal
P	= pressure
Pr	= Prandtl number
Pr_t	= turbulent Prandtl number
q	= heat exchange
Q	= heat release
t	= time
T	= temperature
u, v, w	= velocity components in Cartesian frame
x, y, z	= Cartesian coordinates
γ	= ratio of specific heats
ϵ	= eddy viscosity coefficient
λ	= bulk viscosity coefficient
μ	= molecular viscosity coefficient
ξ, η, ζ	= transformed coordinates
ρ	= density
$\bar{\bar{\tau}}$	= stress tensor

Superscripts

(=)	= tensor
(−)	= vector
(′)	= fluctuation

Introduction

THE numerical simulation of gas flows achieved by solving the compressible Navier-Stokes equations is a relatively new discipline which emerged to meet technological requirements. The basic need exists to find an alternative means from experimental effort to establish or to supplement the data base for aerospace vehicle design at outer limits of the flight envelope where viscous/inviscid interactions usually dominate the flow. Regardless of the detailed and intricate fluid dynamics mechanisms that generate these phenomena, they can still be adequately cataloged as either pressure or vorticity interactions.[1] The most complicated interaction problem associated with a full configuration is a combination of both. Approximate numerical solutions utilizing the simplified governing equations lose their validity because of diminishing local accuracy or failure to completely describe the physics of the investigated problems. Fortunately the three-dimensional, time-dependent, compressible Navier-Stokes equations offer a viable instrument to respond to this challenge. This set of governing equations is the cornerstone of continuum aerodynamics. Its range of validity covers the full spectrum of aircraft and missile applications. Numerous methods have been developed to solve the system of quasilinear partial differential equations. However, in the present effort, only the finite-difference approximations of the compressible Navier-Stokes equations are addressed. Even with this restricted scope, a survey of this magnitude is still prohibitive. Thus the emphasis is further focused on three-dimensional problems. Early research results were limited to simple configurations or the components of a complex aerodynamic formation to delineate the flow structure from a generic viewpoint. An outstanding chronological record of pioneering efforts has been compiled by Peyret and Viviand in 1975.[2] More recently, as large-scale computers have become available, a few complex configuration simulations have been documented. In spite of the painstakingly slow progress in the development of numerical procedures and supporting data processors, significant achievements in the three-dimensional simulation and rapid time variation of fluid dynamics phenomena have been obtained.[3,4] The value of numerical Navier-Stokes simulations has been recognized beyond the realm of research activities.

Dr. J. S. Shang is the Technical Manager of the Computational Aerodynamics Group at the Air Force Wright Aeronautical Laboratory. In this capacity, he leads a group of scientists and engineers who perform basic research in computational aerodynamics for numerical simulation of flow around aerospace vehicles. He received his B.Sc., M.Sc. and Ph.D. degrees from the Cheng Kung University of Taiwan, Polytechnic Institute of New York, and the Ohio State University, respectively. In 1967, Dr. Shang joined the Air Force Aerospace Research Laboratory; he was reassigned to the Flight Dynamics Laboratory, Air Force Wright Aeronautical Laboratories in 1975. Over the past eighteen years, he has made fundamental contributions to computational fluid dynamics, particularly in the areas of inviscid-viscous interaction, self-sustained oscillatory flow, vector processing, and full-scale numerical simulation of aircraft. He has authored numerous technical publications and has lectured at professional meetings and universities. Dr. Shang was the recipient of the 1978 General Foulois Award and the 1983 Primus Award.

Presented as Paper 84-1549 at the AIAA 17th Fluid Dynamics, Plasma Dynamics, and Lasers Conference, Snowmass, Colo., June 25-27, 1984; received July 20, 1984; revision received Dec. 11, 1984.

In the area of practical application, the Navier-Stokes solutions considered are either laminar flow or Reynolds-averaged equations with a turbulence model. According to the classification by Chapman[3] on the successively refined approximation to the full Navier-Stokes equations, they belong to the class III category. In theory, the Navier-Stokes equations really should be implemented only in the area of strong inviscid/viscous interaction with flow separation. For a large class of laminar free-interaction problems where the asymptotic theory yields detailed structure of the interacting boundary layer, the application of Navier-Stokes equations is not even necessary.[5,6] For turbulent flow, however, the basic structure is vastly different and cannot be treated with methods that are simple extensions of laminar interactions. Thus the concept of zonal methods is a logical conclusion. The embedded Navier-Stokes solution region was attempted in the early 1970s by Briley.[7] More recent efforts were also successful using a zonal method to obtain a composite solution for a blunt body at high angle of attack[8,9] and an inclined ogive-cylinder-flare configuration.[10] Significant saving in computer time and storage was clearly illustrated. In general, a prior knowledge of the overall flowfield structure is necessary to achieve an efficient and accurate composite solution procedure. However, in the situation where multiple interacting zones are tightly knit together or an unsteady phenomenon arises, the advantage of the zonal approach over the full equation solution is uncertain. For class IV problems, turbulent eddy simulation, the direct numerical simulation from the complete time-dependent Navier-Stokes equations will permit no simplification. Therefore, as long as we are still in the feasibility-study phase, the use of the Reynolds-averaged equations is fully justified.

Applications have been extended to more and more practical problems. If, however, rapid advancement toward the goal of using this methodology as a design tool is to be achieved, emphasis on research and development requires further focus in specific problem areas; hence the present effort to discuss several critical issues concerning flow simulation by the use of the Navier-Stokes equations. A survey of the open literature serves to summarize our past achievements and identify the need for improvement. Although a unified collection of thoughts and ideas is not anticipated, it is hoped that a balanced perspective of Navier-Stokes simulations will result—and that through the discussion of these issues research activities are stimulated in this vitally important technology.

Finally the future outlook of the Navier-Stokes solution will be projected based on current trends in computer architecture development, the technical need, and the expanding applications of Navier-Stokes equations into interdisciplinary areas of scientific interest.

Governing Equations

In the Eulerian formulation, the conservative equations in integral form for mass, momentum, and energy with respect to a control volume V stationary in the inertial frame and enclosed by the control surface A are

$$\frac{\partial}{\partial t}\iiint \rho \, \mathrm{d}V + \iint \bar{n}\cdot\rho\bar{u}\,\mathrm{d}A = 0 \tag{1}$$

$$\frac{\partial}{\partial t}\iiint \rho\bar{u}\,\mathrm{d}V + \iint \bar{n}\cdot\rho\bar{u}\bar{u}\,\mathrm{d}A = \iint \bar{n}\cdot\bar{\bar{\tau}}\,\mathrm{d}A + \iiint \rho\bar{f}\,\mathrm{d}V \tag{2}$$

$$\frac{\partial}{\partial t}\iiint \rho e\,\mathrm{d}V + \iint \bar{n}\cdot\rho e\bar{u}\,\mathrm{d}A = \iint \bar{n}\cdot(\bar{u}\cdot\bar{\bar{\tau}} - \bar{q})\,\mathrm{d}A + \iiint \rho(\bar{f}\cdot\bar{u} + Q)\,\mathrm{d}V \tag{3}$$

The first equation is the most fundamental principle of Newtonian mechanics; matter can be neither created nor destroyed. The second equation is Newton's second law of motion applied to each differential element of medium within the control volume. The body force per unit mass (acceleration) is designated as $\bar{f}$.

The second equation is also the only vector equation in the system. For a three-dimensional problem, there are three scalar components in the respective degrees of freedom. The principle of conservation of energy, Eq. (3), states that the rate of increase of energy of the medium in the control volume is produced by the flux of energy across the control surface, work done by external causes, and energy generated inside the volume. Here $\bar{q}$ denotes the heat exchange across the control surface, and Q denotes the rate of energy release per unit mass. For the situation concerning chemical reaction, Q is simply the heat of formation of a chemical reaction.

The integral equations hold for any volume element contained in the flowfield. This system of integral equations is the foundation for the finite-volume algorithm[11] and is frequently used in checking the overall validity of approximate numerical solutions. The equivalent differential equation is obtained by a limiting argument in the control volume. In deriving the appropriate differential equation of motion, the differential elements of length, although regarded as negligibly small, must be large enough to permit a molecule to encounter a great many collisions in crossing that length. Since the control volume is stationary, the relative order of time derivative and volume integral may be exchanged. By means of Gauss' divergence theorem, we have

$$\frac{\partial \rho}{\partial t} + \nabla\cdot\rho\bar{u} = 0 \tag{4}$$

$$\frac{\partial \rho\bar{u}}{\partial t} + \nabla\cdot(\rho\bar{u}\bar{u} - \bar{\bar{\tau}}) - \rho\bar{f} = 0 \tag{5}$$

$$\frac{\partial \rho e}{\partial t} + \nabla\cdot(\rho e\bar{u} - \bar{u}\cdot\bar{\bar{\tau}} + \bar{q}) - \rho(\bar{f}\cdot\bar{u} + Q) = 0 \tag{6}$$

If the body force and the energy sink or source within the control volume are omitted and the flow medium is assumed to be incompressible, then the classic Navier-Stokes equations are deducible from the conservation of momentum equations.[12] However, it is now conventional to refer to the complete set of equations of motion as the compressible Navier-Stokes equations, including Fourier's law for heat transfer ($\bar{q} = -k\nabla T$, where k is the molecular thermal conductivity of the flow medium). This system of equations is not closed: there are five equations but ten dependent variables (u, v, w, ρ, p, T, e, k, μ, λ). Additional equations must therefore be included to connect the thermodynamic variables and to describe the proportional coefficients related to the flow medium. For most aerodynamic applications, the equation of state and calorically perfect gas assumption are adopted, together with specification of Sutherland's viscosity law, constant Prandtl number, and zero bulk viscosity ($3\lambda = -2\mu$), to complete the description of the system of equations. For laminar flow, with a set of appropriate initial and boundary conditions, the system of differential equations is solvable in principle. Due to the present limitations of computing facilities, for turbulent flow, one must be satisfied with solutions of the Reynolds-averaged Navier-Stokes equations.

The Reynolds-averaged Navier-Stokes equations are achieved as the ensemble average of the decomposition of dependent variables into a macroscopic and turbulent fluctuation.[13] The equations are written in ensemble average and are valid even in statistically nonstationary flows. For statistically stationary flow, the ensemble average is equivalent to the time average. However, the ensemble average process eliminates several key characteristics of turbulence, for example, the frequency, phase, and wavelength of the fluctuating motion. Fortunately this information is usually not critical for most practical applications. The mass-weight average, due to Favre, is based on the observation that the average mass of fluid contained in a volume bounded by a surface moving with mean mass-weighted velocity is constant.[14] This observation is in perfect accord with Laufer's statement that the density variation has a kinematic or volumetric, rather than a dynamic, effect on the velocity field in a compressible turbulent flow.[15] A careful and systematic study of the turbulent mean-flow, Reynolds-stress, and heat-flux equations in mass-averaged dependent variables was completed by Rubesin and Rose.[16] The main conclusion pertinent to our discussion is that the Reynolds-averaged Navier-Stokes equations are in general identical with the laminar-flow counterparts if one accepts the new definition of stress tensor and heat flux as

$$\bar{\bar{\tau}} = (-p + \lambda \nabla \cdot \bar{u})\bar{\bar{I}} + \mu \,\mathrm{def}\,\bar{u} - \langle \rho \bar{u}' \bar{u}' \rangle \tag{7}$$

$$\bar{q} = -k \nabla T + \langle \rho e' \bar{u}' \rangle \tag{8}$$

where the ensemble average, $\langle\,\rangle$, produces the Reynolds stress and the apparent heat flux. It should be noted, in the above formulation, that the flow is no longer a Newtonian medium, since the Stokes′ hypothesis making the stress linearly proportional to the rate of stress is invalid. Only the application of the eddy viscosity concept will technically reduce the turbulent formulation into the Newtonian.

The open literature, the Reynolds-averaged Navier-Stokes equations are most frequently rewritten in flux vector form for Cartesian coordinates. This coordinate system is always adopted as the basic frame of reference for the successive coordinate transformation to satisfy a particular configuration.

$$\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = 0 \tag{9}$$

The dependent variables U are easily identifiable as ρ, $\rho\bar{u}$, and ρe. The flux vectors F, G, and H are simply the components of the vectorial and tensorial quantities contained within the divergence operator.

$$U = U(\rho, \rho u, \rho v, \rho w, \rho e) \tag{10}$$

$$F = \begin{bmatrix} \rho u \\ \rho u^2 - \tau \\ \rho uv - \tau_{xy} \\ \rho uw - \tau_{xz} \\ \rho eu - \gamma\left(\dfrac{\mu}{Pr} + \dfrac{\epsilon}{Pr_t}\right)\dfrac{\partial e_i}{\partial x} - (u^\tau_{xx} + v^\tau_{xy} + w^\tau_{xz}) \end{bmatrix} \tag{11}$$

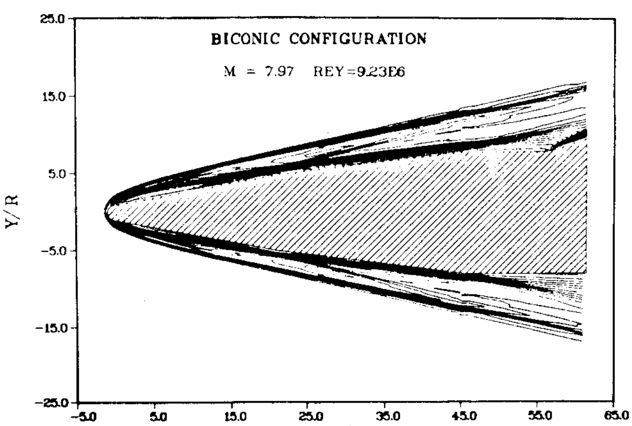

Fig. 1 The entire flowfield of a biconic configuration (density contour), Ref. 42.

$$G = \begin{bmatrix} \rho v \\ \rho uv - \tau_{yx} \\ \rho v^2 - \tau_{yy} \\ \rho vw - \tau_{yz} \\ \rho ev - \gamma\left(\dfrac{\mu}{Pr} + \dfrac{\epsilon}{Pr_t}\right)\dfrac{\partial e_i}{\partial y} - (u^\tau_{yx} + v^\tau_{yy} + w^\tau_{yz}) \end{bmatrix} \tag{12}$$

$$H = \begin{bmatrix} \rho w \\ \rho uw - \tau_{zx} \\ \rho vw - \tau_{zy} \\ \rho w^2 - \tau_{zz} \\ \rho ew - \gamma\left(\dfrac{\mu}{Pr} + \dfrac{\epsilon}{Pr_t}\right)\dfrac{\partial e_i}{\partial z} - (u^\tau_{zx} + v^\tau_{zy} + w^\tau_{zz}) \end{bmatrix} \tag{13}$$

In the above formulation, the eddy viscosity (ϵ) and the definition of the turbulent Prandtl number (Pr_t) were included. This system of equations is identical to those for laminar flow in structure and is probably the simplest form of the Reynolds-averaged Navier-Stokes equations used.

For most engineering applications, Cartesian coordinates are rarely adequate in describing the geometric configuration. There are two reasons for this. First, an extensive interpolation procedure for boundary conditions become necessary. Second, it is difficult to implement a systematic clustering of grid spacing.[17] Thus a generalized coordinate mapping is introduced in the form

$$x = x(\xi,\eta,\zeta), \quad y = y(\xi,\eta,\zeta), \quad z = z(\xi,\eta,\zeta)$$

(A temporal transformation is also possible but is not included in our discussion.) By means of the chain rule of differentiation, the governing equation, Eq. (9), achieves the chain-rule conservation form.[18]

$$\frac{\partial U}{\partial t} + (\xi_x, \xi_y, \xi_z) \begin{vmatrix} \dfrac{\partial F}{\partial \xi} \\ \dfrac{\partial G}{\partial \xi} \\ \dfrac{\partial H}{\partial \xi} \end{vmatrix} + (\eta_x, \eta_y, \eta_z) \begin{vmatrix} \dfrac{\partial F}{\partial \eta} \\ \dfrac{\partial G}{\partial \eta} \\ \dfrac{\partial H}{\partial \eta} \end{vmatrix} + (\zeta_x, \zeta_y, \zeta_z) \begin{vmatrix} \dfrac{\partial F}{\partial \zeta} \\ \dfrac{\partial G}{\partial \zeta} \\ \dfrac{\partial H}{\partial \zeta} \end{vmatrix} = 0 \tag{14}$$

Again, by means of a coordinate transformation identity,[19] the above equations can be rewritten in the strong conservation form[20]

$$\frac{\partial U/J}{\partial t} + \frac{\partial}{\partial \xi}[(\xi_x F + \xi_y G + \xi_z H)/J] + \frac{\partial}{\partial \eta}[(\eta_x F + \eta_y G + \eta_z H)/J] + \frac{\partial}{\partial \zeta}[(\zeta_x F + \zeta_y G + \zeta_z H)/J] = 0 \tag{15}$$

The so-called conservation form in numerical analysis is well known.[21] The conservation form has a telescope effect: The internal cancellation of differencing approximation within a control volume is total for a given differencing approximation of a system of equations with constant coefficients.[21] The relative merits of the different formulations were studied by Hindman on a limited basis.[22] For engineering applications, the difference in accuracy and numerical efficiency is probably not critical.

The description of a differential system of equations is not complete without the specification of initial and boundary conditions. To achieve this objective, knowledge of the appropriate classification of our governing equations is essential. Today we know that the system of time-dependent Navier-Stokes equations is of the incompletely parabolic type.[23,24] For infinite Reynolds number, the system of equations reduces to quasilinear hyperbolic type. At any finite value of Reynolds number, the system is neither elliptic nor parabolic.[25] We will address this subject later in our discussion.

Survey

The survey is grouped into the following categories:
1) Forebody
2) Airfoils and wings
3) Wing-fuselage
4) Afterbody and near wake
5) Propulsion system and combustion
6) Inviscid/viscous interactions
7) Time-dependent problem.

Forebody

Slender-body, blunt-body, projectile, missile-like shape, and planetary-entry configurations are included in this section. The problem area includes a wide variety of body formations: spinning pointed cones,[26-30] ogive cylinders,[31-34] blunt cones,[35-39] blunt biconics with and without flaps,[40-42] general blunt bodies,[43-51] and planetary-entry configurations.[52,53] The methods of solution also use various approximations to the Navier-Stokes equations covering a wide range.[54,55] The forerunner of the recently popularized parabolized Navier-Stokes equations was developed by Cheng[56] and Davis[35] for solving the blunt-body flowfield. The basic idea is to write the nondimensional Navier-Stokes equations in boundary conformal coordinates and compare them with the same set of equations written in variables of order one in the inviscid region. The final composite set of equations is thus uniformly valid for the entire shock layer up to second-order accuracy in terms of the inverse square root of the characteristic Reynolds number.[35] On the other hand, the thin-layer approximation evolved from a realistic assessment based on experience and insight due to Baldwin and Lomax.[57] The thin-layer approximation neglects only the diffusion processes parallel to the body surface which the usually highly stretched mesh parallel to the surface simply cannot resolve. A study of the Navier-Stokes solution and the thin-layer approximation was carried out by Degani and Steger for a two-dimensional flow over a compression ramp.[58] They have found small differences between the numerical results, the discrepancies being minor and confined within the separated flow region. More importantly, in a controlled study for the time-dependent condition, the disparity between the solutions is mainly due to phase shift. Even though this simple configuration poses a serious test for the thin-layer approximation, caution must be exercised in drawing a general conclusion from a single numerical result.

The rigorous mathematical structure of the parabolized Navier-Stokes equations, particularly for the laminar flow, is contained in the framework of the interacting boundary-layer theory. The triple-deck theory by Stewartson and others[5,6] describes clearly the hierarchy of increasing accuracy of approximations to the Navier-Stokes equations. This specialization can be obtained by neglecting the temporal derivatives and the streamwise diffusion within the Navier-Stokes equations, as well as modifying the streamwise convective flux vector to permit the marching of the equations downstream from initial data.[45] Significant computational efficiency and reduced data storage requirements are realized. This simplification, however, should always be applied in conjunction with a boundary conformal coordinate system to gain the maximum advantage. The well-known, ill-posed problem leading to the departure solution and numerical instability under certain flow conditions seems to be controlled by either global iteration or some special numerical procedure.[45,55] This simplified approximation has proved very useful in engineering applications.

The conical approximation has also been applied to the forebody problem to produce a more efficient numerical procedure for pointed-nose configurations. The conical flow structure is maintained by insisting upon the invariance of all flow properties along a generating ray including the body surface considered. For the conical approximation, the intrinsic coordinates are the classical spherical system in that conical invariance requires all the derivatives in the radial direction simply to vanish. In principle, the conical approximation is not able to satisfy the scaling law of the shear-layer growth from its apex. Thus the conical approximation has an inherent deficiency. This discrepancy from physics should be more serious for laminar flows than turbulent flows because of the degree of deviation from the conical structure. However, a detailed study comparing numerical results with flight and wind tunnel data seems to demonstrate that this approximation is a viable and efficient procedure for slender-body problems.[59] In summary for the forebody problem, the current trend is wide usage of the thin-layer and parabolized procedure. The full Navier-Stokes equations were used in early research efforts[35,36] and for a specific study on the vortical pattern at high incidence[31,32] or for a comparative purpose.[42] The shock-fitting procedure advocated by Moretti[60] is most frequently used as an alternative in generating accurately the necessary initial data.

As an illustration, the entire flowfield of a biconic missile-like configuration is given in Fig. 1. The bow shock, the embedded flap-induced shock waves, and the rapid expansions around joint and slices are faithfully reproduced by the Navier-Stokes equations. The detailed pitot pressure profiles on the forebody and the afterbody flap also showed excellent agreement with data[61] (see Fig. 2). This accuracy can be achieved only if the forebody calculation is accurate.

Novel applications of the Navier-Stokes equations and their approximation also extended to spinning bodies with and without[62-64] coning motion.[65] In these related studies, the primary objectives were the resolution of the Magnus force and moment. The consensus points out that the contribution to the Magnus force is equally important among the cross-flow shear, centrifugal force, and displacement interaction effect. Thus, resolution of the viscous/inviscid interaction is critical in order to have any meaningful results for the net Magnus force.

Airfoils and Wings

A major group of the two-dimensional Navier-Stokes solutions in the open literature is devoted to the transonic airfoil problem.[66-80] It is also a naturally self-contained component problem in aircraft applications. The flowfield usually consists of a leading edge, trailing edge, and possibly the coalescing shock waves in the separated flow region over the chord of the airfoil. In order to complete the description of the physical phenomenon, the near and far wake regions are also included. In essence, the airfoil problem is a microcosm of most of the difficulties in numerical simulation one would encounter for the full-scale aircraft investigation. In the early phase of exploratory research in this subject area, an 18% thick biconvex airfoil was adopted for both the numerical simulation and experimental validation.[66-69] Deiwert's work on this airfoil was probably the first effort to use the conservation equation in integral form.[66] Next, Levy initiated the exploratory investigation of the buffet range of an airfoil,[69] and obtained the first time-dependent Navier-Stokes solution for the transonic airfoil. Soon afterward, Steger and Bailey, investigating the transonic aileron buzz problem,[73] found that the inviscid unsteady shock-wave motion is the driving force in transonic aileron buzz. However, the viscous effect is critical and can both sustain and moderate the flap motion. For some conditions, viscous effects even change the frequency and amplitude of the aileron motion. In the same vein, Chyu et al.[75] conducted a comparative study on a NACA 64A010 airfoil oscillation in pitch. Their results seem to indicate that both the inviscid and the thin-layer approximations are in good agreement with experimental data, but the viscous-flow computation is more accurate in predicting air loads in terms of lift and pitching moment coefficients. Little or no information on drag coefficients was documented. Recently a comparative study on finite-difference and integral-differential techniques was conducted by El-Refaee.[78] He found little difference between the two approaches in computer resources used and numerical results obtained at a relatively low Reynolds number of 1000.[78]

In this area of research,[67-82] several issues on numerical simulations emerged concerning the uncertainty of numerical resolution,[79] particularly near the trailing edge of an airfoil where the turbulent structure undergoes an abrupt change and the selection and placement of boundary conditions for the mixed characteristic flowfield are unclear.[80-82] A series of investigations was designed to isolate these uncertainties.[83,84] These research efforts seem to indicate that higher-order closure turbulence models yielded slightly better predictions for the turbulent properties, but a better turbulence model is still needed. It was also determined that the accuracy of results was highly dependent on the degree of numerical resolution of the strong leading- and trailing-edge gradient. Several related research efforts also reflected similar observations.[85-90]

There are only a few solutions of three-dimensional wings known at present. Usually the solving schemes are approximate procedures to the full Navier-Stokes equations whether in the conical or parabolized form. Bluford[91] and Vigneron et al.[92] used the conical approximation to solve delta-wing problems. The shock-induced vortex development above the boundary-layer development was captured and compared favorably with experimental data. The slab delta wing at high angle of attack and hypersonic Mach numbers was simulated by Tannehill et al.[93] with the parabolized Navier-Stokes equations. Their paper also presents an excellent discussion of the departure solution and the treatment of the streamwise pressure gradient. Recently numerical simulations of blunted delta-wing and strake delta-wing configurations were obtained by Fujii and Kutler with the thin-layer approximation.[94] A finite wing (ONERAM6) at transonic Mach number and Reynolds number of 1000 was also completed by Hollanders et al.[95]

Wing-Fuselage

The wing-fuselage problem has a more complex flowfield structure than the finite wing in that additional strong inviscid/viscous interaction also occurs in the wing root region. The interference effect in wing-body junctures is a well-known phenomenon in aircraft[96] and missile aerodynamics.[97,98] Only a few numerical simulations of this configuration are documented.[53,99-101,103] For a relatively complex formation, the memory limitations of today's computers severely restrict the number of discrete nodes that can be accommodated for the calculations. Thus, the choice of coordinate system for a given problem is critical.[53,99,100] In the computation of an ogive forebody and sharp leading-edge delta-wing combination, a cut along the wing surface is essential in obtaining results comparable to that of experiments at a moderate angle of attack.[100] The flowfield is shown in terms of selected density contours at streamwise stations and the oil film patterns on the surface of the wing-body configuration in Figs. 3 and 4. A highly blended forebody-wing configuration and twin-jet afterbody nozzle geometry was simulated by Mace and Cosner.[47,101] The adopted velocity-splitting method[102] in the analysis provides an efficient means of solving the three-dimensional viscous flowfield in subsonic and transonic regions. The flowfield about a finned projectile and missile was accomplished by Rai et al.[103] using the parabolized Navier-Stokes equations. The numerical solution by means of the Navier-Stokes equations was also implemented for the region in which the fins began to protrude in ascertaining the approximate solution. The numerical results obtained for the force coefficients of the finned projectile were found to be in agreement with experimental values.

Afterbody and Near Wake

In this section of discussion, the afterbodies associated with wake and jet plumes are included. The flowfield is usually characterized by an expansion followed by a recompression in the near wake domain. In the case of jet exhaust, a reflective shock-wave system or a single Mach disk structure may result, depending upon the pressure ratio across the jet stream and the ambient condition. Several two-dimensional and axisymmetrical configurations were simulated by solving the full Navier-Stokes equations and their simplified approximation.[104-110] A few three-dimensional flowfields past afterbody and exhaust plumes were also computed successfully,[111-114] by Deiwert and Vatsa et al. The common problems encountered

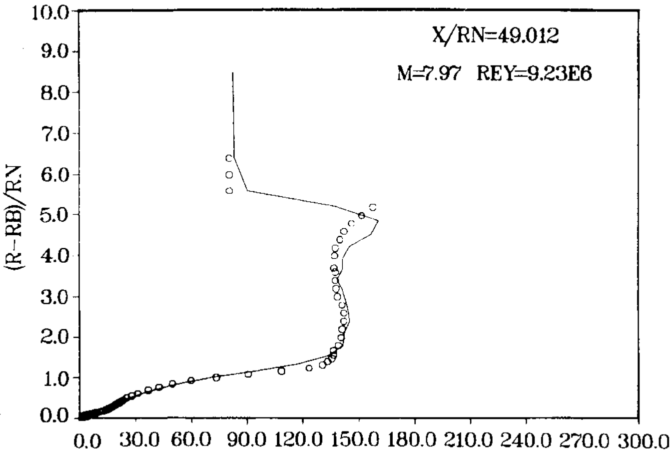

Fig. 2 Comparison of pitot pressure profiles, Ref. 42.

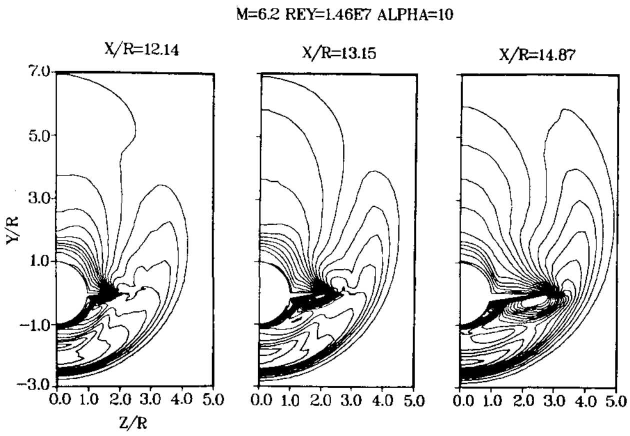

Fig. 3 Density contour around a wing-fuselage configuration, Ref. 100.

in numerical simulations are the solution of an appropriate turbulence model for describing the abrupt flow structure change and its impact on turbulent properties. A unique problem of the afterbody simulation is that the accuracy of the result depends on upstream boundary conditions. A relaxation turbulence model[115,116] was implemented for the near wake study.[106] It was found that the upstream flow history has a profound influence in the subsequent near and far wake region. In spite of the aforementioned concerns, Hasen's work on axisymmetric nozzles clearly demonstrated that the Mach disk reflection is recoverable by the full Navier-Stokes equations.[108] A parabolized Navier-Stokes equation set was also successfully applied in solving a two-dimensional and axisymmetric coflowing jet[107] and three-dimensional exhaust plumes.[112] The novel and efficient procedure is based upon a set of composite equations written in an approximation intrinsic coordinate system.[117]

A series of three-dimensional boattail afterbodies with and without a centered propulsion jet was computed by Deiwert.[111,113,114] His work probably reflects the achievement in the three-dimensional numerical analysis of the afterbody problem by using a thin-layer approximation. Valuable information can be obtained from his effort. First, in using a large array processor, a carefully structured data base is critical in reducing unnecessary data movement and thus the overall expenditure in data processing of a complex configuration. Second, the influence of grid point distribution requires attention. Deiwert has shown that the clustered mesh spacing in the sharp forebody-afterbody junction significantly improved comparison with experiment.[111,118]

Propulsion System and Combustion

Inlets, diffusers, ducts, laser systems, and combustion propulsion systems are contained in the following discussion.[119-133] For supersonic inlets, the strong inviscid/viscous interaction with a multiple shock-wave system is predominant. In particular, the interaction between the shock generated by the inlet cowl lip and the inlet ramp boundary layer triggers a complex reflective wave system in an attempt to achieve wave cancellation. The shock-wave system is usually terminated near the inlet throat by an approximately normal shock. Knight initiated a series of two-dimensional studies in this problem area.[119-121,123] He is probably among the first to incorporate a general grid generation capability with existing algorithms for solution of the Navier-Stokes equations.[119] In this connection, Thompson's elliptic grid generation procedure[124] was used successfully in a strong conservation form. Knight also developed an efficient explicit-implicit procedure for the two- and three-dimensional supersonic inlet problem.[125,126] His numerical results with an eddy viscosity model generally agree with experimental data. However, attention is also drawn toward further development of the turbulence model for multiple-shock-wave/boundary-layer interaction with mass exchange on walls.[126] The parabolized Navier-Stokes approach has also been used for the analysis of high-speed inlets and diffusers. The efforts of McDonald and Briley[127-129] leading to success in the high-speed inlet problem are also noted.[130-133] Of particular interest in this area of research is the significant role played by the numerical resolution and the shock-capturing technique.[131-136] The issue of appropriately posed boundary conditions for subsonic and mixed-type problems which frequently occurs in internal flows continues to be a serious problem.

The duct and diffuser problems have also attracted much research activity.[137-140] The flowfield structure can be made more amiable by using the parabolized Navier-Stokes equations.[137-139] The boundary conditions for the subsonic problem, however, still offer a formidable challenge.[141-143] The work of Yee, Beam, and Warming has assured us at least that stability analysis can provide a rigorous guide to the treatment of numerical boundary conditions for a hyperbolic system of equations.[144,145] For the incompletely parabolic system of equations, one can only produce a strong estimate. Moretti's general concept in specifying physical conditions at infinity for subsonic flows probably should be adhered to.[142] The work of Buggeln et al. in a strongly curved three-dimensional rectangular duct with physical boundary and initial conditions yields the physical structure of the flow correctly.[146] The difficulty of implementing boundary conditions for these internal flows is compensated for by the much better understanding of turbulence. The works of Deardorff and others[147,148] are illustrative.

The Navier-Stokes equations were also used in the combustion propulsion system by Kumar[149] and Drummond and Weidner.[150,151] Novel solutions have been obtained for an internal-combustion engine,[152] and a chemical laser system.[153-155] These research efforts reveal a rapidly growing interdisciplinary thrust of computational aerodynamics in conjunction with chemical kinetics.[156-159]

Inviscid/Viscous Interactions

While solving the inviscid/viscous-interaction problem by using the Navier-Stokes equations is the cornerstone of fluid dynamic research, it is also the building block of all the previous discussions. In the pioneering phase of solving this system of equations, the investigated configurations included the following: cavity,[160] shock tube,[161] blunt body,[162-164] base flows,[165,166] leading edge of a plate,[162] corner,[167] compression ramp,[168] and shock impingement.[169,170] Several excellent reviews[2,171-173] are very good sources for readers interested in the early development. The work of MacCormack signifi-

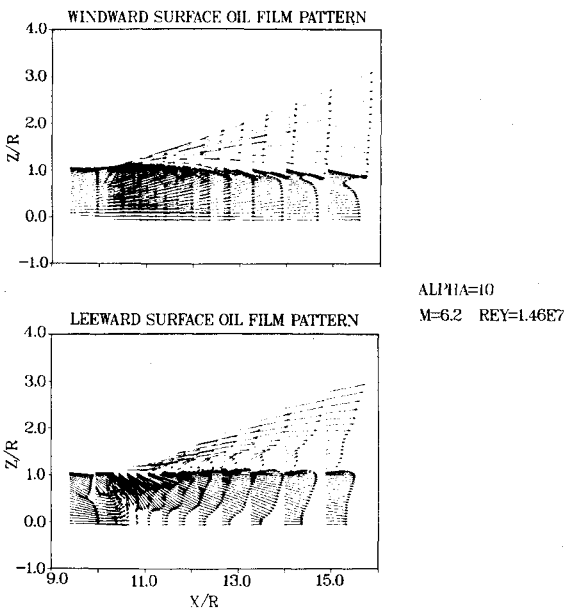

Fig. 4 Shear pattern on the wing-fuselage surface, Ref. 100.

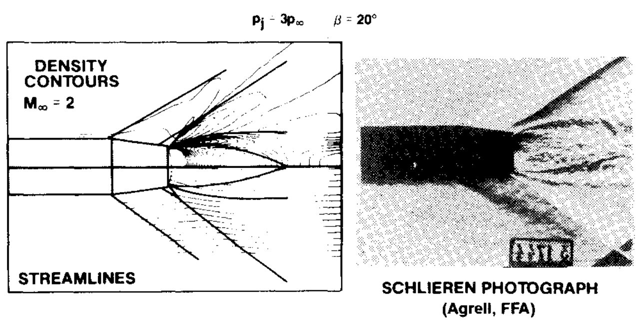

Fig. 5 Comparison of flow structure around a boattail afterbody (Courtesy of G. S. Deiwert).

cantly impacts current activities in solving the Reynolds-averaged Navier-Stokes equations. His efforts in algorithm development[174-176] set the standard for the most active area of numerical research. Most importantly, from the beginning his explicit method has been the necessary tool in solving complex and numerically demanding interacting problems. The meticulous research of Carter[168] also bears the mark of distinction in revealing the link between asymptotic theory and numerical solutions. It would be beyond the present scope of this effort to detail interacting boundary-layer theory, but a good collection of numerical solutions based upon the triple-deck theory can be found in the work of Napolitano et al.[177]

Only shock impingement, compression ramps, corner flow, and the flowfield around a blunt fin are discussed here. To avoid repetition of early documentation,[2,171-173] our discussion is focused on more recent efforts. Tannehill, Rakich, and others studied two-dimensional turbulent, blunt-body flows with an impinging shock wave.[178,179] These efforts demonstrated that the irregular-shaped bow shock can be treated as a discontinuity by shock-fitting. Also, the implicit numerical methods,[180-182] although requiring roughly 1.8 times more computing resources per time step, permitted significant saving in total computing time to complete a given problem because of their favorable stability restriction. The three-dimensional counterpart of the shock-wave impingement problem was solved by Holst.[183] The three-dimensional shock wave impinging on a boundary layer developed internal to a tube (Kussoy et al.[184]) and on a body of revolution (Brosh et al.[185]) was studied side by side with experimental efforts, and reasonable agreement was reached with the accompanying data (Fig. 9). A jet impingement problem relevant to vertical takeoff and landing aircraft design was conducted by Agarwal and Bower[186] with a two-equation turbulence model.[187] They prefer the use of the higher-order closure model for turbulence. However, no specific comparison was presented in this work.

The inviscid/viscous interactions over a compression ramp at supersonic and hypersonic Mach number were continuously studied and compared with experiments[188-195] (Fig. 10). The numerical results generated by the simple flux-gradient concept gave reasonable agreement with experiments for the location of separation and the overall pressure rise. Significant

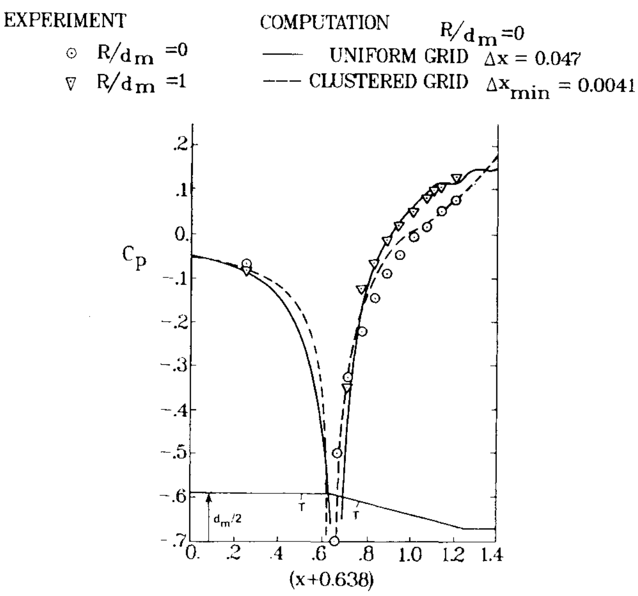

Fig. 6 Influence of grid distribution on afterbody pressure distribution, Ref. 111 (Courtesy of G. S. Deiwert).

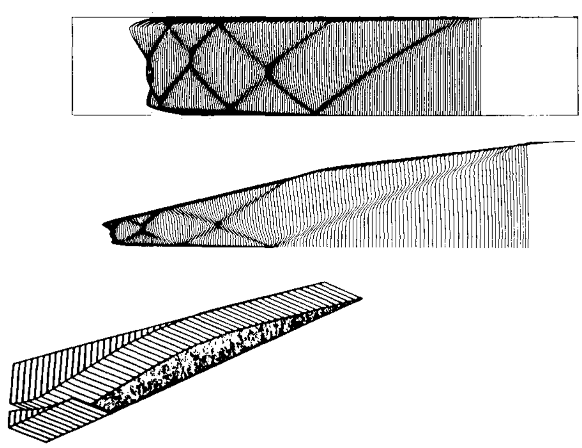

Fig. 7 Shock-wave system of a high-speed inlet, Ref. 131 (Courtesy of B. Anderson).

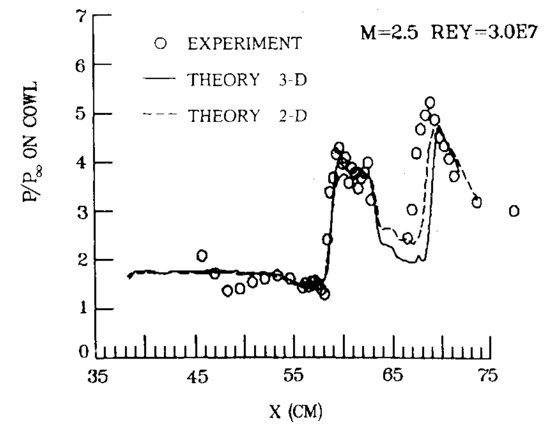

Fig. 8 Surface pressure distribution on cowl of an inlet, Ref. 126 (Courtesy of D. D. Knight).

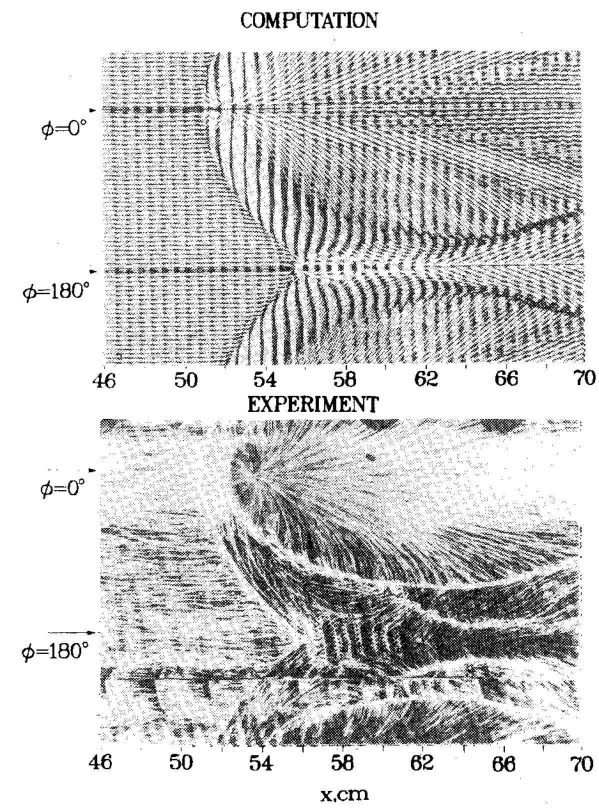

Fig. 9 Comparison of surface shear distributions on a body of revolution with an impinging shock, Ref. 185 (Courtesy of C. M. Hung).

discrepancies occur only in the separated flow and particularly in the reattachment region.[190-192] The higher-order closure for turbulence[196-199] seems to provide a slightly better prediction. The consensus also favors the turbulence model by Wilcox and Rubesin.[197,198] The contributions by Horstman, Settles, Bogdonoff, Viegas, Coakley, and Rubesin in this series of studies are significant. They have also isolated the cause of disparity in numerical prediction, turbulence models, and the area of improvement needed. The requirement for turbulence model refinement is obvious but, significantly, the essential features of the physical phenomenon were predicted adequately by the numerical analysis. The three-dimensional flows around the flared juncture of an inclined body of revolution was investigated by Hung and Chaussee. Hung solved the thin-layer approximation equation for the laminar flow case.[200] In his analysis the flare can be viewed as an axisymmetric compression ramp encountering the oncoming stream with incidence. The numerical results show good agreement with experimental measurements of surface pressure and normal force distribution (Fig. 11). He also demonstrated that the circumferential communication through the cross flow plays an important role in the three-dimensional shock-wave/boundary-layer interaction. As a natural extension, a turbulent case was considered under a composite solution procedure including the full Navier-Stokes equations, the thin-layer, and the parabolized approximation.[10] They demonstrated that the thin-layer approximation yielded comparable accuracy (4%) and significant reduction in computer resources (25%) to that of the full Navier-Stokes equations.[10,200] Impressive numerical efficiency is also exhibited by the parabolic approximation. Again, the deficiency of the turbulence model in the leeward flowfield prediction was reported.

Corner flow is one of the most complex three-dimensional inviscid/viscous interacting phenomena.[201,202] In the absence of shock waves, the numerical solutions based on the method of matched asymptotic expansion were obtained by Ghia and Davis,[203] and Mikhail and Ghia.[204] A numerical study was also performed by Li.[205] The flowfield becomes very complex when the shock-wave system is generated either by the geometric configuration or the growth of a boundary layer. Frequently a triple point will form in the region where waves intersect each other.[206,207] The turbulence described by a simple algebraic model with an appropriate-length scale yields correctly the shortening scale in the interacting zone (see Fig. 12). Hung, MacCormack, and Horstman[208-211] also studied corner-flow problems characterized by an overwhelming single shock-wave system over the corner region. They not only demonstrated the improved numerical efficiency but also the accuracy of the thin-layer approximation. Regardless of the detailed mechanism from which the cross flow was created,[212,213] the high heat-transfer rate adjacent to the corner is always associated with the thinning of the wall shear layer. A flowfield external to an axial corner was also investigated by Kutler, Pulliam, and Vigneron.[214]

Another group of inviscid/viscous interacting problems, those involving a blunt fin mounted on a plate, has attracted intensive investigation for the primary horseshoe and secondary vortex structure.[11,215,216] The numerical solution[11,216] showed that indeed it can effectively supplement the experimental effort for aerodynamic design as well as for the understanding of basic fluid dynamics phenomenon. Additional information regarding three-dimensional separated flow and validation data of inviscid/viscous interaction can be found in the works of Tobak and Peake[217] and Kline et al.,[218] respectively.

Time-Dependent Problem

Our fascination with the unsteady aerodynamics problem is more than natural curiosity. The majority of engineering problems to be solved are unsteady, as once the flowfield becomes turbulent it is by definition a time-dependent phenomenon.[219] Since our current computing facilities simply cannot process data in the Kolmogrov scales,[220] the class of problems we hope to simulate is restricted to flow containing organized motion. Unfortunately it is impossible, for this class of problems, to distinguish whether the organized periodic motion is superimposed on a background of turbulence or, perhaps more accurately, vice versa.[221] In the numerical simulation of the unsteady problem with the Reynolds-averaged equations, the ambiguity of implementing a turbulence model at high oscillating frequencies is still an unresolved question; the classic problem encountered in turbulence research is trying to understand energy spectra and explaining why most of the energy contribution comes from the low wave number components of the fluctuating motion.[221] While this issue is still open to question, persistent numerical studies, together with long-term fundamental research efforts, may eventually provide some much-needed understanding.

The pioneering research efforts to duplicate the von Kármán vortex street by Payne,[222] Fromm,[223] and others[224-226] are very well known. Continual efforts[227-229] are still being undertaken to better understand this fundamental fluid dynamics phenomenon. The unsteady airfoil problem[230] and related aileron buzzing problem are also being investigated.

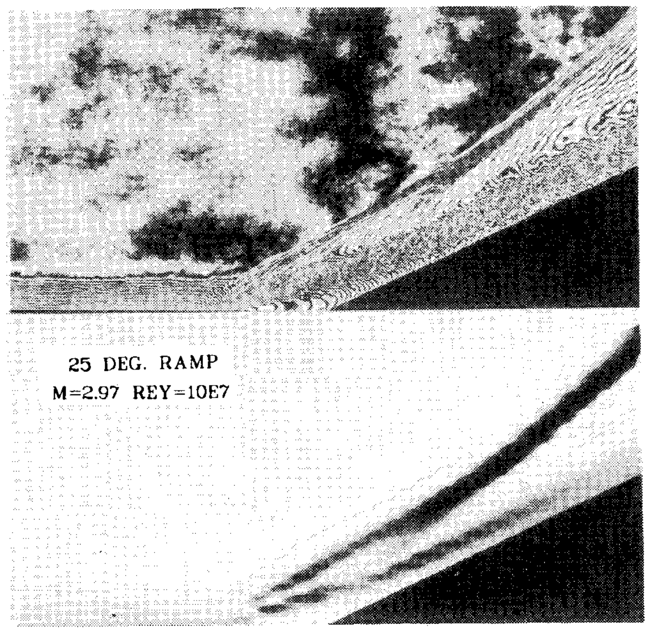

Fig. 10 Supersonic flow over a compression ramp.

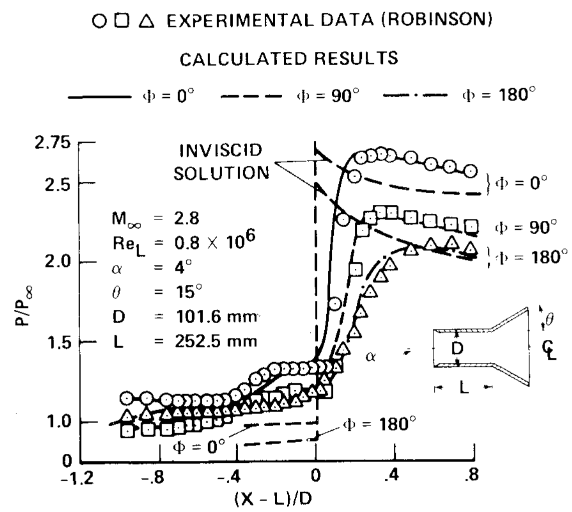

Fig. 11 Comparison of surface pressure distributions on the flared juncture, Ref. 200 (Courtesy of C. M. Hung).

The early numerical results obtained by Levy[69] and by Steger and Bailey[73] have been discussed previously and will not be repeated here. In engineering applications, the free-shear-layer impingement type of self-sustained oscillatory flow[231-233] is the most frequently encountered. This type of oscillatory fluid motion must have a feedback mechanism to complete the chain of events. First, the free-shear-layer instability selectively amplifies disturbances within a certain range of frequencies; then the free shear layer impinges on a surface, and the disturbance is reintroduced into the system. The feedback mechanism usually presents itself in the form of a pressure pulse and propagates upstream through the subsonic portion of the flowfield. The pressure oscillation in an open cavity[234] and the spike-tipped or indented nosetips problems[235-238] belong to this group. The numerical results by Shang, Hankey, and Smith compared favorably with experimental data not only in predicted frequency and amplitude but also in the detailed waveform[239] (see Fig. 13).

Oscillatory flow in the form of inlet buzz,[240] unsteady transonic flow in a diffuser[241] and combustor[242] have been successfully simulated by the time-dependent Navier-Stokes equations. An interesting application was extended to a three-dimensional turret problem[243] and its wake structure controlled by mass suction.[244] Of particular interest is that the comparison of rms density fluctuation across the shear layer with experimental results reveals a general agreement using only a simple algebraic turbulence model, while the highest numerically resolvable frequency is around 6 kHz. Hankey's contribution in developing a better understanding of self-sustained oscillatory fluid motion is invaluable.[232,234,238] Numerical study of the hydrodynamic instability leading to transition was also conducted by using the incompressible Navier-Stokes equations[245,246] and their compressible counterparts.[247] For the compressible simulation, in spite of uncertainty in numerical resolution and the imposition of appropriate boundary conditions, the predicted discrete frequencies at various streamwise locations seem to indicate a clear agreement with experimental observation. It is clear that additional efforts are required to ascertain numerical simulations in this research area. The reviewer feels that the domain of direct calculation of turbulence through large eddy simulation is critically important for the long-term understanding of turbulence. It is also a highly specialized area of research and is considered to be outside the scope of discussion. However, several current state-of-the-art review articles by Leonard, Ashurst, and Morchoisne are included.[248-250]

Assessments

The following assessments on numerical simulation address the areas of emphasis for further growth.

Efficiency and Accuracy of Numerical Procedures

Since 1960[2,21] numerous finite-difference algorithms have been developed for solving the compressible Navier-Stokes equations. Today, however, in the open literature the most frequently used algorithms for solving this system of equations can be identified as MacCormack's algorithms, [174-76] Beam and Warming's approximate factorization scheme,[182] Briley and McDonald's procedure,[128,181] and several hybrid methods.[125,251,252] There are also some more recent developments.[253-256] The multigrid technique[257] and spectral methods[258,259] have also been introduced into the area of solving the Navier-Stokes equations.[260] The multigrid technique is basically a numerical procedure which systematically suppresses the short wave components of numerical error residuals by using varying mesh systems, thus enhancing the rate of iterative convergence. The fundamental requirement is to condition the matrix structure of the governing equations and to transfer the solution smoothly from one grid-point system to the other. The potential acceleration of convergence by the use of multigrid is obvious. Some success in practical applications has been reported by Chima and Johnson[261] and Davis.[262] The use of spectral methods is based on the fact that for a smoothly varying function, the series approximation, either by the Fourier or Chebschev series, gives accuracy for derivatives far superior to the common finite-difference procedure. For a given accuracy criterion the spectral method would require fewer grid points than would finite-differencing methods. However, to develop this concept for practical application, substantial efforts are still needed.[246,262-265] At present we have too small a data base to make a resonable assessment of its efficiency and accuracy in solving the compressible Navier-Stokes equations.

For the most common finite-differencing algorithms several general statements may be given. In general, the implicit procedure requires more arithmetic operations per time step than its explicit counterpart. However, the more favorable stability characteristics and a potentially faster convergence rate of the implicit procedure more than compensates for the overhead in the matrix inversion process to yield a substantial saving in computing resources. Unfortunately a general index cannot be easily established because the specific comparison is always problem-dependent as well as dependent on the accuracy requirement. Finally, the most stringent accuracy evolution of numerical solutions using the scaling law of the triple-deck theory was accomplished by Hussaini et al.[266] In theory, the scientific precision is achievable with the aforementioned numerical procedures.

The hybrid procedures were developed from the observation that in most solutions the aspect ratio of grid spacings among the streamwise, peripheral, and normal directions is substantial, due to the highly stretched mesh distribution normal to the surface. Over the computational domain it is not unusual to observe the CFL time-step size spanning a range of several

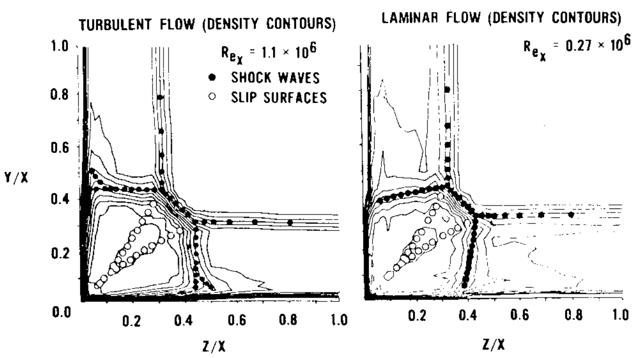

Fig. 12 Comparison of turbulent and laminar corner flows, Ref. 207.

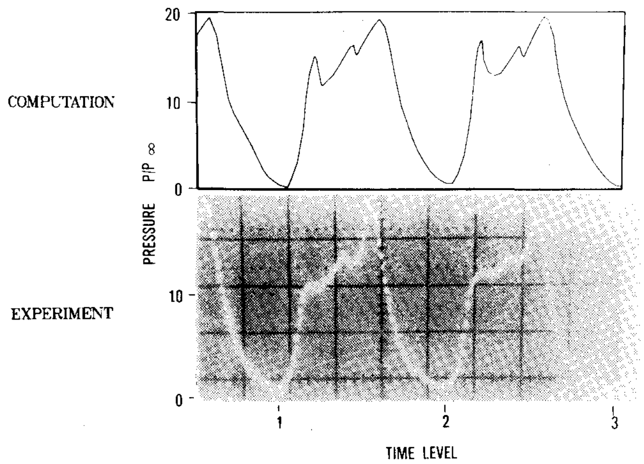

Fig. 13 Comparison of pressure waveform of an oscillating spike-tipped body, Ref. 239.

decades. One group of hybrid procedures alleviates the stability constraint by implementing an implicit subdomain within the framework of an explicit scheme.[125,175,251] Another class of hybrid procedures is aimed at eliminating the implicit operation in the directions in which the favorable stability property of the differencing operator is no longer advantageous.[252] These hybrid numerical procedures have demonstrated high numerical efficiency.

Other procedures intended to improve computational efficiency can be identified as convergence-enhancement, specialization, higher-accuracy, and computer-adapted procedures. The multigrid procedure, spectral method, and local time-step method belong to the first group. The spectral method, particularly the collocation approach, can bring boundary values into the field instantaneously. For this reason alone, it can be considered a device of convergence enhancement. However, the pseudospectral approximation in a three-dimensional calculation indicates a better accuracy than a fourth-order finite-difference approximation.[267] The local time-step procedure is really the simplest convergence enhancement for problems with an asymptotic steady state. The distorted time-step procedure simply satisfies the stability condition at each point individually. The solution therefore progresses at different temporal rates at each point toward the asymptote. This procedure was first used by Li for a three-dimensional Navier-Stokes calculation.[36] Typically an increase of two to five times in convergence to a steady-state problem is indicated.

Specialization plays a significant role in reducing computing resources by utilizing simplified governing equations and zonal methods. The former is reflected in work on thin-layer approximation[57] and the parabolized equations.[45] While both methods reduce the computational resources required, the parabolized procedure is a space-marching method, additionally saving substantial computer storage. However, the zonal method used in an attempt to produce a sophisticated far-field boundary condition has met with no outstanding success as yet. The problem of determining the criterion of equation switching and the compatibility of the interphase condition remains unresolved.

In the area of higher-order accuracy reside the traditional higher-order procedures[253,268]: spectral, higher-order spline,[269,270] and adaptive-grid methods.[271,272] For the adaptive-grid method, the fundamental idea is to redistribute grid points in the region of high gradient, thus reducing the large local truncation error. The accuracy improvement is offset, however, by additional calculation and interrogation expenditure. For very large-scale calculations, the coordinate transformation derivatives or the direction cosines of the control surface are usually selectively computed as the need arises to avoid large storage space. Thus, the repetitive update of the grid-point system is not an overly restrictive constraint. However, the implementation of this procedure and the optimal criterion of adaptation still needs additional research effort.

Computer adaptation is probably a major factor in recent advancement in improving computing efficiency. Code preparation requires the programmer to know much more about the computer architecture and its particular characteristics. It is not unusual to achieve an order of magnitude or more increase in data-processing rate between a high-speed scalar computer and a class VI vector processor with a theoretical calculation rate of 50 to 80 million floating point operations per second (FLOPS).[273,274] For an explicit algorithm, vectorization is achieved by simply insisting on a better program and close attention to the data structure. However, for an implicit method, due to the recursive relation in the matrix inversion operation, vectorization is inhibited (backscatter uncertainty). A new way of matrix solving is required. Pulliam and Lomax[48] and Benek et al.[275] overcame this difficulty and successfully vectorized a three-dimensional code based upon the Beam-Warming algorithm.

From the previous discussion, a trend in algorithm development is clearly revealed. The process is laboriously evolutional after the initial conceptual breakthrough. The algorithm and numerical procedure will remain a major focal point of research activity in computational fluid dynamics. Then a natural selection process will take place, in which the most widely used numerical procedure will be the one that yields accurate results and has rapid convergence characteristics, ease of programming, and reliability.

Well-Posed and Stable Boundary Conditions

There is no rigorous mathematical theorem concerning the proper initial and boundary conditions to ensure existence and uniqueness of the solution[276] to the Navier-Stokes equations. However, our experience in solving this system of equations tells us to impose the condition that takes into account the physical meaning of the problem and the mathematical nature of the system of equations. This physical requirement of a mathematical formulation is referred to as the well-posed problem.[171] A well-posed problem can be achieved only if the boundary conditions are properly specified. The solutions of the differential equation then vary continuously with respect to perturbations of the initial and boundary conditions. Successful numerical solutions must also possess this property, namely, that slight perturbation of the initial and boundary conditions yield a correspondingly small perturbation of the numerical result.

In order to develop a well-posed problem, a definition of the type of partial differential equation being considered is essential. Unfortunately our knowledge of classification of coupled, partial differential equations is very limited. Even for a partial differential equation of second order with more than two independent variables, it is not always possible to reduce the equation to a simple canonical form.[277] More recent research efforts by Belov and Yanenko,[23] Strikwerda,[24] and Gustafasson and Sundstrom,[25] have identified the time-dependent Navier-Stokes equations as being of the incompletely parabolic type. For certain, in the asymptotic limit of Reynolds number approaching infinity, the time-dependent Navier-Stokes equations reduce to a quasilinear hyperbolic system. For any finite value of Reynolds number, the system is no longer hyperbolic, but neither is it elliptic or parabolic.

The incompletely parabolic system of partial differential equations is basically a mixed initial boundary value problem. Some limited general results of the well-posedness conditions for incompletely parabolic systems were derived from classical energy methods.[25] In this mode of analysis, the system is improperly posed if there are eigensolutions that grow at an arbitrarily large rate and are thus unbounded. From these analyses, one may only conclude that for most aerodynamic problems there are no unique boundary conditions known in advance except for flow within solid wall boundaries, where the boundedness of the energy norm is always satisfied.[25] In addition, the proper number of independent boundary conditions is uniquely determined by the rank of the coefficient matrix of the partial differential equations and the number of negative eigenvalues of the diagonalized matrix.

In the numerical analysis, the implementation of stable numerical boundary conditions is also a major issue and one that is not completely resolved.[144,145] The problem arises from a computational domain that is bounded in space and in time. It is common practice to introduce artificial boundaries[21] to fit the investigated problem into the limited memory of the computer. It is also well known that most finite-difference methods need more boundary conditions than the differential equations. For a stable and accurate numerical algorithm improper implementation of boundary conditions (either analytical or numerical) can lead to instability and inaccuracy. Thus, the construction of stable and convenient conditions at artificial boundaries becomes critical. Unfortunately this is a difficult area of research, as defining and obtaining stability bounds is hindered by mathematical and conceptual complexities.

In spite of formidable obstacles, wide-ranging numerical solutions of the Navier-Stokes equations have been obtained. These results usually compare favorably with the accompanying experimental observations. This criterion of validation is fully justified based upon the fact that the Navier-Stokes equations were developed to describe physical phenomena. In engineering applications, these results yield insight and understanding unattainable by other means. To date, we can compute the Reynolds-averaged Navier-Stokes equations routinely in the supersonic domain for both the time-dependent and steady-state problems.

In subsonic numerical simulations, no clear pattern of success is apparent. As an example, for the simulation of subsonic flow over a cylinder at a nominal Mach number of 0.5, three different sets of boundary conditions were used in the far field: the extrapolation condition, the condition derived from the characteristic variables,[278] and the Strikwerda type of no-reflection condition.[279] The pressure distributions along the axis of symmetry of the cylinder exhibit indifferent behavior with respect to the different boundary conditions and the placement of the boundary. The trapped wave behavior is obvious on the surface pressure distribution, and no repeatable pressure wave pattern is achieved over an extensive period of time. However, the extrapolation condition and overspecified boundary conditions were successfully applied to a three-dimensional turret problem.[243,244] The only plausible explanation that can be offered is that the three-dimensional relief effect permits the overspecification of boundary conditions. For internal flow problems such as inlets and diffusers, the notable successes in specification of boundary conditions have been a combination of physical conditions and extrapolation.[133] Usually this includes the description of the stagnation pressure and temperature at the entrance and the static pressure at the exit, while the rest of the boundary conditions are defined by extrapolation.

In the transonic flow regime, numerous Navier-Stokes solutions for flow over airfoil have been obtained. The flowfield structure is characterized by a relatively small perturbation. Even though there is no rigorous study of the decay rate of this disturbance, the far-field boundary is typically placed several chords away from the source of the perturbation. The boundary conditions were a mixed group of physical and extrapolation conditions. In this class of problems, the range of eigenvalues is widely scattered, and the studied phenomenon is frequently time-dependent.[69,70,74,75,79,82] Again, as in the subsonic numerical simulation, the data base of successful specification of far-field boundary conditions for the Navier-Stokes equations is rather limited.

It is evident that the ability to establish a well-posed and stable boundary condition for solving the Navier-Stokes equations is paramount. A physically meaningful solution must have realistic physical boundary conditions implemented in an unambiguous manner. Intense research efforts require a synergistic approach with experimental and mathematical disciplines. From the experimental research a better-defined problem and validation is essential. From mathematic discipline, solid guidance must be established to perform systematic numerical investigations.

Turbulence Model

Understanding turbulence phenomena is one of the few scientific frontiers of aerodynamic research. Numerous definitions of turbulence have been given, but the characteristics of the turbulence pertaining to aerodynamics can be summarized as follows: Turbulence is a random, continuum, and strictly three-dimensional phenomenon. Turbulence is a highly dissipative and diffusive process which, most importantly, is a property of the flow at high Reynolds number but not the flow medium. In a field of fluctuating flow, many length and velocity scales coexist.[219] The smallest scale of motion is governed by the value of molecular viscosity through the dissipative process. These fine scales are commonly referred to as the Kolmogrov microscales or the inner scales.[220] In order to resolve the smallest eddy size, the number of nodes of a numerical analysis reaches an astronomical value.[3] In aerodynamics, turbulence is always associated with a solid boundary and confined closely to the wake or jet region. This region usually is very limited in dimension, but its influence on the evaluation of aerodynamic force and moment is profound. However, based upon our understanding of the dissipative process of turbulence, the large-scale motion contains most of the turbulent energy and performs the momentum transport. In applications, the question of how fine a scale of motion needs to be resolved for a given problem probably has to await research activities in large-scale eddy simulation.[3,248-250]

There is no contradicting the fact that the Navier-Stokes equations contain the necessary information for turbulence and even the laminar-turbulent transition. Even though we have no direct and positive result to demonstrate this fact, enough outstanding research efforts offer promise. There is a gap between scientific understanding and practical applications. However, significant efforts were made to bridge the gap by the use of the transport equations. Recent works by Bradshaw et al.,[280] Reynolds,[281] and Marvin[282] have elegantly summarized these efforts and hence will not be repeated here.

The mean turbulent field closure models are the result of an approximation in describing the generation tensor, pressure-strain redistribution tensor, and the dissipative tensor of the turbulent transport equations. Frequently a differential equation for length scale is also incorporated. The detailed approaches are clearly beyond the scope of the present effort. All these higher-order closure models seems to yield a slightly better prediction than the algebraic approximation for a certain class of problems.[191-194,282] Since turbulence is the property of a particular flowfield, it would be unrealistic to expect that there is a universal turbulence model with a fixed set of universal constants. Finally, in spite of sophistication in the turbulence model for the Reynolds stress, the turbulent heat flux is simply evaluated by a constant value of the turbulent Prandtl number. Therefore, it would not be surprising if most three-dimensional Navier-Stokes calculations adopted algebraic turbulence models.[57,283]

The algebraic turbulence model, in theory, has a very limited range of validity and is, at best, a crude description of turbulence. However, in engineering applications, the simple flux-gradient concept exceeds expectation. The degree of success in practical applications is primarily due to the dissipative characteristics of turbulence. It is very forgiving. The algebraic turbulence model may provide a means of capturing the key and global features of a flowfield. However, it is not reliable in yielding fine details of secondary features. Therefore, if the secondary feature is the basic mechanism in a chain of events, failure is to be anticipated. In general, these conditions prevail in flows of high extra-strain rate, lacking a dominant axis of strain, and instability enhanced by viscous effects. Specific examples are catastrophic flow separation, near wake, flow associated with extremely high angles of attack, shear driven internal peripheral flow, and a class of self-sustained internal oscillatory flows. However, these difficulties may not be uniquely limited to turbulent simulations by the algebraic model.

It is obvious that turbulence modeling will remain indefinitely as a critical issue to be resolved. Realizing the fundamental limitation, a proper perspective seems to suggest that concentrated efforts should be put into a class of problems sharing common characteristics. The side-by-side approach with experiment will generate a solid base for a directly usable result for engineering applications. The need is urgent.

Grid Generation and Data Structure

For very large-scale data processing the mutual dependence between grid generation and data structure becomes increasingly important for efficient operation,[114,273,284] particularly if the concept of multicomputational domains is to be implemented or the data accession has a particular bias. It is common in three-dimensional calculations for the portion of computing resources consumed in transferring data between units to equal the computing resource required to execute the arithmetic operations. Once external memory is used for storage, the resources required to ship data can be overwhelming.[114] For an efficient numerical simulation, a carefully constructed data structure designed specifically for the numerical procedure and computer architecture is critical.

The appropriate selection of an optimal coordinate system for a given problem is paramount. It is not unheard of for the success or failure of a solution to hinge on the first step of the numerical procedure.[90,100] The general criterion for a suitable grid system is usually vague and is part of the preparatory work of a numerical simulation, allowing a lot of imagination and creativity. Therefore, it always suffers from lack of standardization. Unfortunately this is the first requirement of a truly objective evaluation of the soundness of a grid system. At present, grid generation gradually becomes one of the major expenditures in terms of calendar time for a given numerical simulation.

In general, current technology in grid generation can be classified into two major groups according to the basic approaches to this problem. On the one hand, grid generation can be achieved by solving partial differential equations, but it can also be developed by algebraic interpolation between boundaries. An outstanding collection of works in numerical grid generation was prepared by Thompson,[17] fittingly reflecting his illuminative contribution in this area of research.[285] There is no substitute for studying the original works of each author, but the relative merits of the groups of grid generation can be summarized. The obvious advantage of grid generation by conformal mapping[286] and solutions of partial differential equations[124,287,288] is that the resultant grid system is continuous and differentiable to the order of the differential equation adopted. Since grid generation by solving the partial differential equation is a field method, the degree of control is governed by the nature of the differential equations. The orthogonal preference and the gradual grid-spacing stretching rate are systematically obtainable with a minimum of user effort. For the algebraic interpolation procedures,[289,290] the values of the coordinates in the field are determined by interpolation formulas. Control of the transformed coordinate is explicitly accomplished through stretching functions in the formulation. The smoothness of a grid generated algebraically is not always ensured for piecewise continuous contour, and user effort may be more intensive than with field methods. However, the algebraic procedure usually requires few computational resources and is more amenable to the modular construction of complex configurations.

Complex three-dimensional configurations present additional challenges for problem solving in two regards: 1) the topological constraints imposed by the complicated shape and multiconnected domains, and 2) the internal compatibility of a grid system containing highly contrasting geometric definitions.

Finally, in the area of grid generation, one perceives that the fundamental tools are available for the designated task. However, if the present trend persists, the grid-generation phase of numerical simulation will occupy a major portion of calendar time in problem solving. There is immense room for innovative ideas, but some of the bench tools should be standarized in order to form a common library.

Post Processing of Data and Data Display

Post processing of data and data display is the most neglected problem areas in solving the Navier-Stokes equations. In the research phase of development, a project is considered complete once validation of the solution is obtained by comparing with experimental data, analytic results, or similar numerical simulation. A vast amount of the information generated was usually discarded after major points were made. Storage and manipulation of the huge amount of data became a serious problem. In the process of developing the comparison, the data must first be extracted and then compressed together in tabular form or in graphic display. For the latter form of presentation, creativity and imagination were required to achieve the clearest depiction. For this reason, pockets of excellent graphic capability existed at different institutions with state-of-the-art graphics work stations.[291,292] At present, a wide spectrum of color graphics terminals is gradually being made available in the interactive mode. The color graphics display possesses value beyond aestheticism. The pseudo color spectrum can reveal detailed flow structure which traditional presentation has failed to disseminate. Color graphics is an important new tool for presenting the computed result and making it easier to understand.

The post processing of three-dimensional and time-dependent problems severely taxes the data storage system. This is particularly true for the time-dependent problem, where time-contiguous data must be stored and then manipulated to produce the desired result. In order to understand the investigated phenomenon, a dynamic display of data is necessary. By nature this is a multidisciplinary endeavor, coordinating special skills required to achieve a visual end product, thus making the synergistic effect more apparent. The graphic representation of three-dimensional problems is confronted by a fundamental limitation: The projection of a three-dimensional image onto a plane frequently confuses rather than clarifies the intended results. The remedy is available in optical physics through holographic technology. Currently this new optical display is relatively expensive for routine use, but the progress in this research area is impressive. Fortunately the requirement for computer graphics is not a lonely voice in our technical effort.

In the application of the Navier-Stokes equations for aircraft design, the final result needed is the aerodynamic force and moment. For the experimental effort, the integrated information is readily available by the balancing instrumentation. In numerical simulations, additional effort is needed to derive this information from the discretized field. This linkage is provided by Cauchy's theorem, $\bar{F}=\bar{n}\bar{\bar{\tau}}$. Where the shear tensor contains the static pressure, the $\bar{n}$ is simply the outward normal of the body. Again, the advantage of using boundary conformal coordinates is apparent. The net aerodynamic force exerted on the configuration is obtainable using the area integral.

Future Prospect

From our previous discussion, we have seen no inherent difficulties that would inhibit further development in solving the Navier-Stokes equations efficiently and accurately. The rapid progress of technical support in the form of high-speed computing facilities and graphics system will accelerate the technology transition from research to application. Most importantly, over the short span of a decade and a half, the number of researchers in this area has grown from a handful to thousands. The objectives of applying the compressible, three-dimensional Navier-Stokes equations to aircraft design is clearly achievable. The projections provided by Chapman,[3] Kutler,[4] and Korkegi[293] are scientifically sound and probably conservative. We have seen problems solved that were judged to be impossible just a short while ago. Historically the peril of predicting the impossible is great.

The trend of technical needs seems to suggest a greater challenge and risk in the area of interdisciplinary work. The combination of fluid dynamics with structure, flight dynamics, chemical kinetics, optical physics, and electromagnetic physics, to name a few, is predictable. The shorter-term promising technical opportunities can be identified from aircraft design requirements.[294] They can be singled out easily as transonic drag prediction, vortical flows,[295,296] compressors,[297] deep-stalled subsonic airfoils,[222] and the time-dependent problems.

One feels confident that within the next few years a full-scale numerical simulation of aircraft will be realized. In the next decade, computational aerodynamics will supplement the test matrix of the designer's data basis. Even with the silicon technology in computer components, one fails to detect any major stumbling blocks to provide a fully interactive design system for aircraft and missile. Further, we will be in a position to achieve the optimum for the design activities mentioned earlier. In the area of turbulence research, advancements in solution procedures and facilities will enable us to take strides in understanding that has eluded us for more than a century.

Acknowledgments

The author gratefully acknowledges the stimulating discussions provided by Wilbur L. Hankey Jr., Donald P. Rizzetta, Jeff E. Graham, and members of the Computational Aerodynamics Groups. He is also deeply indebted to Valerie J. Holler for help in preparing this manuscript.

References

[1]Hayes, W. D. and Probstein, R. E., *Hypersonic Flow Theory,* Academic Press, New York, 1959, pp. 333-374.

[2]Peyret, R. and Viviand, H., "Computation of Viscous Compressible Flows Based on the Navier-Stokes Equations," AGARDograph No. 212, 1975.

[3]Chapman, D. R., "Computational Aerodynamics Development and Outlook," *AIAA Journal,* Vol. 17, No. 12, 1979, pp. 1293-1313.

[4]Kutler, P., "A Perspective of Theoretical and Applied Computational Fluid Dynamics," AIAA Preprint 83-0037, Jan. 1983.

[5]Stewartson, K., "Multistructed Boundary Layers on Flat Plates and Related Bodies," *Advances in Applied Mechanics,* Vol. 14, 1974, pp. 145-239.

[6]Adamson, T. C. Jr. and Messiter, A. F., "Analysis of Two-Dimensional Interactions Between Shock Wave and Boundary Layers," *Annual Review of Fluid Mechanics,* Vol. 12, 1980, pp. 103-138.

[7]Briley, W. R., "A Numerical Study of Laminar Separation Bubbles Using the Navier-Stokes Equations," *Journal of Fluid Mechanics,* Vol. 47, 1971, pp. 713-736.

[8]Chaussee, D. S., Patterson, J. L., Kutler, P., Pulliam, T. H., and Steger, J. L., "A Numerical Simulation of Hypersonic Viscous Flows Over Arbitrary Geometries at High Angle of Attack," AIAA Preprint 81-0050, 1981.

[9]Rizk, Y. M., Chaussee, D. S., and McRae, D. S., "Computation of Hypersonic Viscous Flow Around Three-Dimensional Bodies at High Angle of Attack," AIAA Preprint 81-1261, 1981.

[10]Hung, C. M. and Chaussee, D. S., "Computation of Supersonic Turbulent Flows Over an Inclined Ogive-Cylinder-Flare," *AIAA Journal,* Vol. 19, No. 9, Sept. 1981, pp. 1139-1144.

[11]Hung, C. M. and Kordulla, W., "A Time-Split Finite-Volume Algorithm for Three-Dimensional Flow-Field Simulation," AIAA 6th Computational Fluid Dynamics Conference Proceedings, Danvers, Mass., July 13-15, 1983, pp. 494-504.

[12]Navier, M., "Memoire sur les Lois du Mouvements des Fluides," *Memoires de l'Academie des Sciences del'Institut de France,* Vol. 6, 1827.

[13]Reynolds, O., "On the Dynamical Theory of Incompressible Viscous Fluids and the Determination of the Criterion," *Philosophical Transactions of the Royal Society of London, Series A,* Vol. 186, 1895, p. 123.

[14]Favre, A., "Equations des Gaz Turbulents Compressible," *Journal de Mecanique,* Vol. 4, No. 3, Sept. 1965, pp. 361-390.

[15]Laufer, J., "Thoughts of Compressible Turbulent Boundary Layers," Symposium on Compressible Turbulent Boundary Layers, Langley Research Center, Dec. 10-11, 1968, NASA SP-216.

[16]Rubesin, M. W. and Rose, W. C., "The Turbulent Mean-Flow Reynold-Stress, and Heat Flux Equations in Mass-Averaged Dependent Variables," NASA, TM X-62, 248, NASA Ames Research Center, March 1973.

[17]Thompson, J. F., ed., *Numerical Grid Generation,* North-Holland, New York, Amsterdam, Oxford, 1982.

[18]Shang, J. S. and Hankey, W. L., "Numerical Solution of the Navier-Stokes Equations for a Three-Dimensional Corner," *AIAA Journal,* Vol. 15, No. 11, Nov. 1977, pp. 1575-1582.

[19]Viviand, H., "Conservative Forms of Gas Dynamics Equations," *La Recherche Aerospatiale,* No. 1, Jan.-Feb. 1974, pp. 65-68.

[20]Pulliam, T. H. and Steger, J. L., "Implicit Finite Difference Simulation of Three-Dimensional Compressible Flow," *AIAA Journal,* Vol. 18, No. 2, Feb. 1980, pp. 159-167.

[21]Roache, P. J., *Computational Fluid Dynamics,* Hermosa Publishers, Albuquerque, New Mexico, 1972, pp. 70-72.

[22]Hindman, R. G., "Generalized Coordinate Forms of Covering Fluid Equations and Associated Geometrically Induced Errors," *AIAA Journal,* Vol. 20, No. 10, Oct. 1982, pp. 1359-1367.

[23]Belov, Y. U., Y. A., and Yanenko, N. N., "Influence of Viscosity of the Smoothness of Solutions of Incompletely Parabolic Systems," *Math Notes,* 10, 1971, pp. 480-483.

[24]Strikwerda, J. C., "Initial Boundary Value Problems for Incompletely Parabolic Systems," PhD Thesis, Department of Mathematics, Stanford University, Stanford, Calif., 1976.

[25]Gustafasson, B. and Sandstrom, A., "Incompletely Parabolic Problems in Fluid Dynamics," SIAM *Journal of Applied Mathematics,* Vol. 35, No. 2, Sept. 1978, pp. 343-357.

[26]Lin, T. C. and Rubin, S. G., "Viscous Flow Over a Cone at Moderate Incidence," *Computers and Fluids,* Vol. 1, No. 1, Jan. 1973, pp. 37-57.

[27]Lubard, S. C. and Helliwell, W. S., "Calculation of the Flow on a Cone at High Angle of Attack," *AIAA Journal,* Vol. 12, July 1974, pp. 965, 974.

[28]Helliwell, W. S. and Lubard, S. C., "An Implicit Method for Three-Dimensional Viscous Flow with Application to Cones at Angle of Attack," *Computers and Fluids,* Vol. 3, No. 1, 1975, pp. 83-101.

[29]McRae, D. S., "A Numerical Study of Supersonic Viscous Cone Flow at High Angle of Attack," AIAA Paper 76-77, June 1976.

[30]Lin, A. and Rubin, S., "Three-Dimensional Supersonic Viscous Flow Over a Cone at Incidence," *AIAA Journal,* Vol. 20, No. 11, Nov. 1982, pp. 1500-1507.

[31]Hankey, W. L., Graham, J. E., and Shang, J. S., "Navier-Stokes Solution of a Slender Body of Revolution at Incidence," *AIAA Journal,* Vol. 20, No. 6, June 1982, pp. 776-781.

[32]Graham, J. E. and Hankey, W. L., "Computation of the Asymmetric Vortex Pattern for Bodies of Revolution," *AIAA Journal,* Vol. 21, No. 11, Nov. 1983, pp. 1500-1504.

[33]Nietubicz, C. J., Pulliam, T. H., and Steger, J. L., "Numerical Solution of the Azimuthal-Invariant Thin-Layer Navier-Stokes Equations," *AIAA Journal,* Vol. 18, No. 12, Dec. 1980, pp. 1411-1412.

[34]Degani, D. and Schiff, L. B., "Computation of Supersonic Viscous Flows Around Pointed Bodies at Large Incidences," AIAA Paper 83-0034, Jan. 1983.

[35]Davis, R. T., "Numerical Solution of the Hypersonic Viscous Shock Layer Equations," *AIAA Journal,* Vol. 8, No. 5, May 1970, pp. 843-851.

[36]Li, C. P., "Numerical Solution of Viscous Reacting Blunt Body Flows of a Multicomponent Mixture," AIAA Paper 73-302, June 1973.

[37]Li, C. P., " A Numerical Study of Laminar Flow Separation on Blunt Flared Cones at Angle of Attack," AIAA Paper 74-585, June 1974.

[38]Srivastave, M. J., Werle, J. J., and Davis, R. T., "Viscous Shock-Layer Solutions for Hypersonic Sphere Cones," *AIAA Journal,* Vol. 16, No. 2, Feb. 1978, pp. 137-144.

[39]Kumar, A., Graves, R. A. Jr., and Tiwari, S. N., "Laminar and Turbulent Flow Over a Spherically Blunt Cone with Massive Surface Blowing," *AIAA Journal,* Vol. 17, No. 12, Dec. 1979, pp. 1326-1331.

[40]Helliwell, W. S., Dickinson, R. P., and Lubard, S. C., "Viscous Flow Over Arbitrary Geometries at High Angle of Attack," *AIAA Journal,* Vol. 19, No. 2, Feb. 1981, pp. 191-197.

[41]Gnoffo, P. A., "Hypersonic Flows Over Biconic Using a Variable-Effective-Gamma, Parabolized-Navier-Stokes Code," AIAA Paper 83-1666, July 1983.

[42]Shang, J. S. and MacCormack, R. W., "Flow Over a Biconic Configuration with an Afterbody Compression Flap-A Comparative Numerical Study," AIAA Paper 83-1668, July 1983.

[43]Viviand, H. and Ghazzi, W., "Numerical Solution of the Compressible Navier-Stokes Equations at High Reynolds Numbers with Application to the Blunt Body Problem," *Lecture Notes in Physics* Vol. 59, 1976, pp. 434-439.

[44]Hsie, H. T., "Calculation of Viscous Hypersonic Flow Over a Severely Indented Nosetip," AIAA Paper 83-0721, Jan. 1983.

[45]Schiff, L. B. and Steger, J. L., "Numerical Simulation of Steady Supersonic Viscous Flow," *AIAA Journal*, Vol. 18, No. 12, Dec. 1980, pp. 1421-1430.

[46]Li, C. P., "Implicit Solution for the Shock-Layer Flow Around General Bodies," *AIAA Journal*, Vol. 20, No. 2, Feb. 1982, pp. 175-183.

[47]Cosner, R. R., "Relaxation Solution for Viscous Transonic Flow About Fighter-Type Forebodies and Afterbodies," AIAA Paper 82-0252, Jan. 1982.

[48]Pulliam, T. H. and Lomax, H., "Simulation 3-D Compressible Viscous Flow on the Illiac IV Computer," AIAA Paper 79-0206, Jan. 1979.

[49]Chaussee, D. S., Patterson, J. L., Kulter, P., Pulliam, T. H., and Steger, J. L., "Numerical Simulation of Hypersonic Viscous Flows Over Arbitrary Geometries at High Angle of Attack," AIAA Paper, Jan. 1981.

[50]Kutler, P., Pedelty, J. A., and Pulliam, T. H., "Supersonic Flow Over Three-Dimensional Ablated Nosetips Using an Unsteady Implicit Numerical Procedure," AIAA Paper 80-0063, 1983.

[51]Chaussee, D. S., Buning, P. G., and Kirk, D. B., "Convair 990 Transonic Flow Field Simulation About the Forward Fuselage," AIAA Paper 83-1785, July 1983.

[52]Gnoffo, P. A., "A Finite-Volume, Adaptive Grid Algorithm Applied to Planetary Entry Flow Fields," *AIAA Journal*, Vol. 21, No. 9, Sept. 1983, pp. 1249-1254.

[53]Venkatapathy, E., Rakich, J. V., and Tannehill, J. C., "Numerical Solution of Space Shuttle Orbiter Flow Field," AIAA Paper 82-0028, Jan. 1982.

[54]Davis, R. T. and Rubin, S. G., "Non-Navier-Stokes Viscous Flow Computations," *Computers and Fluids*, Vol. 8, No. 1, 1980, pp. 101-132.

[55]Rubin, S. G., "A Review of Marching Procedures for Parabolized Navier-Stokes Equations," *Proceedings of a Symposium on Numerical and Physical Aspects of Aerodynamic Flows*, Springer-Verlag, New York 1982, pp. 171-186.

[56]Cheng, H. K., "The Blunt-Body Problem in Hypersonic Flow at Low Reynolds Number," Cornell Aeronautical Laboratory, Rept. AF-1285-A-10, 1963.

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

[58]Degani, D. and Steger, J. L., "Comparison Between Navier-Stokes and Thin-Layer Computations for Separated Supersonic Flow," *AIAA Journal*, Vol. 21, No. 11, Nov. 1983, pp. 1604-1606.

[59]Peake, D. J., Fisher, D. F., and McRae, D. S., "Flight, Wind Tunnel and Numerical Experiments with a Slender Cone at Incidence," *AIAA Journal*, Vol. 20, No. 10, Oct. 1982, pp. 1338-1345.

[60]Moretti, G., "On the Matter of Shock Fitting," *Proceedings of the Fourth International Eng. of Numerical Methods in Fluid Dynamics*, Boulder, Colo., June 1974, pp. 287-292.

[61]Coutler, S. M. and Carver, D. B., "Heat Transfer Surface Pressure and Flow Field Surveys on the AEDC Biconic with Slices and Flaps at Madi 8-MAT Phase 1," AEDC-TR-81-V24, July 1981.

[62]Lin, T. C. and Rubin, S. G., "Viscous Flow Over Spinning Cones at Angle of Attack," *AIAA Journal*, Vol. 12, No. 7, July 1974, pp. 975-984.

[63]Sturek, W. B. and Schiff, L. B., "Numerical Simulation of Steady Supersonic Flow Over Spinning Bodies of Revolution," *AIAA Journal*, Vol. 20, No. 12, Dec. 1982, p. 1724.

[64]Agarwal, R. and Rakich, J. V., "Hypersonic Laminar Viscous Flow Past Spinning Cones at Angle of Attack," *AIAA Journal*, Vol. 20, No. 4, April 1982, pp. 479-487.

[65]Agarwal, R. and Rakich, J. V., "Supersonic Laminar Viscous Flow Past a Cone at Angle of Attack in Spinning and Coning Motion," *AIAA Journal*, Vol. 20, No. 6, June 1982, pp. 761-768.

[66]Deiwert, G. S., "Numerical Simulation of High Reynolds Number Transonic Flows," *AIAA Journal*, Vol. 13, No. 10, Oct. 1975, pp. 1354-1359.

[67]McDevitt, J. B., Levy, L. L. Jr., and Deiwert, G. S., "Transonic Flow About a Thick Circula-Arc Airfoil," *AIAA Journal*, Vol. 14, No. 5, May 1976, pp. 606-613.

[68]Deiwert, G. S., "Computation of Separated Transonic Turbulent Flows," *AIAA Journal*, Vol. 14, No. 6, June 1976, pp. 735-740.

[69]Levy, L. L. Jr., "Experimental and Computational Steady and Unsteady Transonic Flows About a Thick Airfoil," *AIAA Journal*, Vol. 16, No. 6, June 1978, pp. 564-572.

[70]Steger, J. L., "Implicit Finite-Difference Simulation of Flow About Arbitrary Two-Dimensional Geometries," *AIAA Journal*, Vol. 16, No. 7, July 1978, pp. 679-686.

[71]Walitt, L., King, L. S., and Lin, C. Y., "Computation of Viscous Transonic Flow About a Lifting Airfoil," *AIAA Journal*, Vol. 16, No. 8, Aug. 1978, pp. 775-776.

[72]Seegmiller, H. L., Marvin, J. G., and Luy, L. L. Jr., "Steady and Unsteady Transonic Flow," *AIAA Journal*, Vol. 16, No. 12, pp. 1262-1270.

[73]Steger, J. L. and Bailey, H. E., "Calculation of Transonic Aileron Buzz," *AIAA Journal*, Vol. 18, No. 3, March 1980, pp. 249-255.

[74]Shamroth, S. J. and Gilbeling, H. J., "Navier-Stokes Solution of the Turbulent Flow Field About an Isolated Airfoil," *AIAA Journal*, Vol. 18, No. 12, Dec. 1980, pp. 1409-1410.

[75]Chyu, W. J., Davis, S. S., and Chang, K. S., "Calculation of Unsteady Transonic Flow Over an Airfoil," *AIAA Journal*, Vol. 19, No. 6, June 1981, pp. 684-690.

[76]Marvin, J. G., Levey, L. L. Jr., and Seegmiller, H. L., "Turbulence Modeling for Unsteady Transonic Flows," *AIAA Journal*, Vol. 18, No. 5, May 1980, pp. 489-496.

[77]El-Rafaee, M. M., Wu, J. C., and Lekondis, S. G., "Solution of Compressible Navier-Stokes Equations Using the Integral Method," *AIAA Journal*, Vol. 20. No. 3, March 1982, pp. 356-362.

[78]El-Rafaee, M. M., "Comparison Between the Integro-Differential Technique and the Finite Difference Method in Solving Unsteady Compressible Viscous Flow Over Airfoils," AIAA Paper 83-1912, July 1983.

[79]Chyu, W. L. and Kuwahara, K., "Computations of Transonic Flow Over an Oscillating Airfoil with Shock-Induced Separation," AIAA Paper 82-350, Jan. 1982.

[80]Tassa, Y. and Sankar, N. L., "Dynamic Stall of an Oscillating Airfoil in Turbulent Flow Using Time Dependent Navier-Stokes Solver," *Unsteady Turbulent Shear Flow*, Springer-Verlag, New York, 1981, pp. 185-196.

[81]Sankar, N. L. and Tassa, Y., "Reynolds Number and Compressibility Effects on Dynamic Stall of a NACA 0012 Airfoil," AIAA Paper 80-0010, Jan. 1980.

[82]Tassa, Y. and Shuster, D., "Navier-Stokes Solution of Shock Boundary-Layer Interactions at Transonic Speed," 2nd Symposium on Numerical and Physical Aspects of Aerodynamic Flows, Jan. 17-20, 1983, California State University, Long Beach, Calif.

[83]Johnson, D. A., Horstman, C. C., and Bachalo, W. D., "Comparison Between Experiment and Prediction for a Transonic Turbulent Separated Flow," *AIAA Journal*, Vol. 20, No. 6, June 1982, pp. 737-744.

[84]Horstman, C. C., "Prediction of Separated Asymmetric Trailing Edge Flows at Transonic Mach Numbers," *AIAA Journal*, Vol. 21, No., 9 Sept. 1983, pp. 1255-1261.

[85]Patel, V. C. and Scheuever, G., "Calculation of Two-Dimensional Near and Far Wakes," *AIAA Journal*, Vol. 20. No. 7, July 1982, pp. 900-907.

[86]Viswanath, P. R. and Brown, J. L., "Separated Trailing Edge Flow at a Transonic Mach Number," *AIAA Journal*, Vol. 21, No. 6, June 1983, pp. 801-807.

[87]Coakley, T. J., "Turbulence Modeling Methods for the Compressible Navier-Stokes Equations," AIAA Paper 83-1693, July 1983.

[88]Rose, W. C. and Sigimer, A., "Calculation of Transonic Flow over Supercritical Airfoil Sections," *Journal of Aircraft*, Vol. 15, No. 8, Aug. 1978, pp. 514-519.

[89]Chen, Z. D. and Bradshaw, P., "Calculations of Viscous Transonic Flow Over Airfoils," AIAA Paper 82-0997, Jan. 1982.

[90]Rhie, C. M. and Chow, W. L., "Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation," *AIAA Journal*, Vol. 21, No. 11, Nov. 1983, pp. 1525-1532.

[91]Bluford, G. S. Jr., "Numerical Solution of the Supersonic and Hypersonic Viscous Flow Around the Delta Wings," *AIAA Journal*, Vol. 17, No. 9, Sept. 1979, pp. 942-949.

[92]Vigneron, Y. C., Rakich, J. V., and Tannehill, J. C., "Calculation of Supersonic Viscous Flow Over Delta Wings With Sharp Subsonic Leading Edges," AIAA Paper 78-1137, July 1978.

[93]Tannehill, J. C., Venkatapathy, E., and Rakich, J. V., "Numerical Solution of Supersonic Viscous Flow Over Blunt Delta Wings," *AIAA Journal*, Vol. 20, No. 2, Feb. 1982, pp. 203-210.

[94]Fujii, K. and Kutler, P., "Numerical Simulation of the Leading Edge Separation Vortex for a Wing and Strake-Wing Configuration," AIAA Paper 83-1908, July 1983.

[95]Hollanders, H., Lerat, A., and Peyret, R., "3-D Calculation of Transonic Viscous Flows by an Implicit Method," AIAA Paper 83-1953, July 1983.

[96]Ashly, H. and Rodden, W. P., "Wing-Body Aerodynamic Interaction," *Annual Review of Fluid Mechanics*, Vol. 4, 1972, pp. 431-472.

[97]Nielsen, J. N., *Missile Aerodynamics,* McGraw-Hill, New York, 1960.

[98]Nielsen, J. N., "Problem Associated with the Aerodynamic Design of Missile Shapes," 2nd Symposium on Numerical and Physical Aspects of Aerodynamic Flows, Jan. 17-20, 1983, California State University, Long Beach, Calif.

[99]Shang, J. S., "Numerical Simulation of Wing'Fuselage Interference," *AIAA Journal,* Vol. 20, No. 12, Dec. 1982, pp. 1657-1658.

[100]Shang, J. S., "Numerical Simulation of Wing-Fuselage Aerodynamic Interaction," AIAA Paper 83-0225, Jan. 1983.

[101]Mace, J. L. and Cosner, R. R., "Analysis of Viscous Transonic Flow Over Aircraft Forebodies and Afterbodies," AIAA Paper 83-1366, June 1983.

[102]Dodge, P. R., "Numerical Method for 2D and 3D Viscous Flows," *AIAA Journal,* Vol. 15, No. 7, July 1977, pp. 961-965.

[103]Rai, M. M., Chaussee, D. S., and Rizk, Y. M., "Calculation of Viscous Supersonic Flows Over Finned Bodies," AIAA Paper 83-1667, July 1983.

[104]Mikhail, A. G., Hankey, W. L., and Shang, J. S., "Computation of a Supersonic Flow Past an Axisymmetric Nozzle Boattail with Jet Exhaust," *AIAA Journal,* Vol. 18, No. 8, Aug. 1980, pp. 869-875.

[105]Peery, K. M. and Forester, C. K., "Numerical Simulation of Multistream Nozzle Flows, *AIAA Journal,* Vol. 18, No. 9, Sept. 1980, pp. 1088-1093.

[106]Waskiewicz, J. D., Shang, J. S., and Hankey, W. L., "Numerical Simulation of Near Wakes Utilizing a Relaxation Turbulence Mode," *AIAA Journal,* Vol. 18, No. 12, Dec. 1980, pp. 1440-1445.

[107]Vatsa, V. N., Werle, M. J., and Anderson, O. L., "Solution of Slightly Underexpanded Two-Dimensional and Axisymmetric Coflowing Jets," *AIAA Journal,* Vol. 19, No. 3, March 1981, pp. 303-310.

[108]Hasen, G. A., "Navier-Stokes Solutions for an Axisymmetric Nozzle," *AIAA Journal,* Vol. 20, No. 9, Sept. 1982, pp. 1219-1227.

[109]Shrewsbury, G. D. and Tassa, Y., "Numerical Simulation of Transonic Flow About Isolated Afterbodies," AIAA Paper 83-0498, Jan. 1983.

[110]Swanson, R. C., Rubin, S. G., and Khosla, P. K., "Calculation of Afterbody Flows with a Composite Velocity Formulation," AIAA Paper 83-1736, July 1983.

[111]Deiwert, G. S., "Numerical Simulation of Three-Dimensional Boattail Afterbody Flowfields," *AIAA Journal,* Vol. 19, No. 5, May 1981, pp. 582-588.

[112]Vatsa, V. N., Werle, M. J., Anderson, O. L., and Hankins, G. B. Jr., "Solution for Three Dimensional Over or Underexpanded Exhaust Plumes," *AIAA Journal,* Vol. 20, No. 9, Sept. 1982, pp. 1188-1194

[113]Deiwert, G. S., "A Computational Investigation of Supersonic Axisymmetric Flow Over Boattails Containing a Centered Propulsive Jet," AIAA Paper 83-0462, 1983.

[114]Deiwert, G. S., "Three-Dimensional Flow Over a Conical Afterbody Containing a Centered Propulsive Jet: A Numerical Simulation," AIAA Paper 83-1709, July 1983.

[115]Shang, J. S. and Hankey, W. L. Jr., "Numerical Solution for Supersonic Turbulent Flow Over a Compression Ramp," *AIAA Journal,* Vol. 13, No. 10, Oct. 1975, pp. 1368-1374.

[116]Shang, J. S., Hankey, W. L. Jr., and Law, C. H., "Numerical Simulation of Shock Wave-Turbulent Boundary Layer Interaction," *AIAA Journal,* Vol. 14, No. 10, Oct. 1976, pp. 1451-1457.

[117]Anderson, O. L., "Calculation of Internal Viscous Flows in Axisymmetric Ducts at Moderate to High Reynolds Numbers," *International Journal of Computers and Fluids,* Vol. 8, 1980, pp. 391-411.

[118]Shrewsbury, G. D., "Effect of Boattail Juncture Shape in Pressure Drag Coefficients of Isolated Afterbodies," NASA TM X-1517, 1968.

[119]Knight, D. D., "Numerical Simulation of Realistic High-Speed Inlets Using the Navier-Stokes Equations," *AIAA Journal,* Vol. 15, No. 11, Nov. 1977, pp. 1583-1589.

[120]Knight, D. D., "Improved Calculation of High Speed Inlet Flows: Part I: Numerical Algorithm," *AIAA Journal,* Vol. 19, No. 1, Jan. 1981, pp. 34-41.

[121]Knight, D. D., "Improved Calculation of High Speed Inlet Flows: Part II: Results," *AIAA Journal,* Vol. 19, No. 2, Feb. 1981, pp. 172-179.

[122]Chaussee, D. S. and Pulliam, T. H., "Two-Dimensional Inlet Simulation Using a Diagonal Implicit Algorithm," *AIAA Journal* Vol. 19, No. 2, Feb. 1981, pp. 153-159.

[123]Knight, D. D., "Calculations of High Speed Inlet Flows Using the Navier-Stokes Equations," *Journal of Aircraft,* Vol. 18, 1981, pp. 748-754.

[124]Thompson, J., Thames, F., and Mastin, C., "Automatic Numerical Generation of Body-Fitted Curvilinear Coordinate System for Field Containing any Number of Arbitrary Two-Dimensional Bodies," *Journal of Computational Physics,* Vol. 15, July 1974, pp. 299-319.

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

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

[127]McDonald, H. and Briley, W. R., "Three-Dimensional Supersonic Flow of a Viscous or Inviscid Gas," *Journal of Computational Physics,* Vol. 19, Oct. 1975, pp. 150-178.

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

[129]Briley, W. R. and McDonald, H., "On the Structure and Use of Linearized Block Implicit Schemes," *Journal of Computational Physics,* Vol. 34, 1980, pp. 54-72.

[130]Buggeln, R., McDonald, H., Kreskovsky, J., and Levy, R., "Computation of Three-Dimensional Viscous Supersonic Flow in Inlets," AIAA Paper 80-0194, Jan. 1980.

[131]Anderson, B. and Towne, C., "Numerical Simulation of Supersonic Inlets Using a Three-Dimensional Viscous Flow Analysis," AIAA Paper 80-0194, Jan. 1980.

[132]Benson, T. and Anderson, B., 'Validation of a Three-Dimensional Viscous Analysis of Axisymmetric Supersonic Inlet Flow Fields," AIAA Paper 83-0540, Jan. 1983.

[133]Liu, N. S., Shamroth, S. J., and McDonald, H., "Numerical Solution of the Navier-Stokes Equations for Compressible Turbulent Two/Three Dimensional Flows in the Terminal Shock Region of an Inlet/Diffuser," AIAA Paper 83-1892, July 1983.

[134]Thomas, P. D. and Lombard, C. K., "Geometric Conservation Law and its Application to Flow Computations on Moving Grids," *AIAA Journal,* Vol. 17, No. 10, Oct. 1979, p. 1030-1037.

[135]Riklis, R. P. and Thomas, P. D., "Shock-Capturing Algorithm for the Navier-Stokes Equations," *AIAA Journal,* Vol. 20, No. 9, Sept. 1982, p. 1212.

[136]Presley, L., "A Comparison of a Shock-Capturing Technique with Experimental Data For Three-Dimensional Internal Flows," Aerodynamic analyses required advanced computers, NASA, SP 349, 1975, pp. 623-642.

[137]Briley, W. R., "Numerical Method for Predicting Three Dimensional Steady Viscous Flows in Ducts," *Journal of Computational Physics,* Vol. 14, 1974, pp. 8-28.

[138]Roberts, D. and Forester, C., "Parabolic Procedure for Flows in Ducts with Arbitrary Cross Section," *AIAA Journal,* Vol. 17, No. 1, Jan. 1979, pp. 33-40.

[139]Patankar, S. V. and Spalding, D. B., "A Calculation Procedure for Heat, Mass, and Momentum Transfer in Three-Dimensional Parabolic Flow," *International Journal of Heat and Mass Transfer,* Vol. 15, 1972, pp. 1787-1806.

[140]Patankar, S. V., Pratap, V. S., and Spalding, D. B., "Prediction of Laminar Flow and Heat Transfer in Helically Wiled Pipes," *Journal of Fluid Mechanics,* Vol. 62, 1974, pp. 539-551.

[141]Cline, M. C., "Stability Aspects of Diverging Subsonic Flow," *AIAA Journal,* Vol. 18, No. 5, May 1980, pp. 534-539.

[142]Moretti, G. and Pandolfi, M., "Critical Study of Calculations of Subsonic Flows in Ducts," *AIAA Journal,* Vol. 19, No. 4, April 1981, pp. 449-457.

[143]Griffin, M. D. and Anderson, J. D. Jr., "On the Application of Boundary Conditions to Time Dependent Computations for Quasi-One-Dimensional Fluid Flows," *Computers and Fluids,* Vol. 5, 1977, pp. 127-137.

[144]Yee, H. C., Beam, R. M., and Warming, R. F., "Boundary Approximations for Implicit Schemes for One-Dimensional Inviscid Equations of Gasdynamics," *AIAA Journal,* Vol. 20, No. 9, Sept. 1982, pp. 1203-1211.

[145]Beam, R. M., Warming, R. F., and Yeek, H. C., "Stability Analysis of Numerical Boundary Conditions and Implicit Difference Approximations for Hyperbolic Equations," NASA CP 2201, Numerical boundary condition procedure, Oct. 1981, pp. 257-202.

[146]Buggeln, R. C., Briley, W. R., and McDonald, H., "Solution of the Navier-Stokes Equations for Three-Dimensional Turbulent Flow with Viscous Sublayer Resolution," AIAA Paper 81-1023, June 1981.

[147]Deardorff, J. W., "A Numerical Study of Three-Dimensional Turbulent Channel Flow at Large Reynolds Numbers," *Journal of Fluid Mechanics,* Vol. 41, 1970, pp. 453-480.

[148]Reitman, V., Isaeli, M., and Wolfshtein, M., "Numerical Solution of the Reynolds Stress Equations in a Developing Duct Flow," AIAA Paper 83-1883, July 1983.

[149]Kumar, A., "Two-Dimensional Analysis of a Scramjet Inlet Flowfield," *AIAA Journal,* Vol. 20, No. 1, Jan. 1982, pp. 96-97.

[150]Drummond, J. P. and Weidner, E. H., "Numerical Study of a Scramjet Engine Flowfield," *AIAA Journal,* Vol. 20, No. 9, Sept. 1982, pp. 1182-1187.

[151]Weidner, E. H. and Drummond, J. P., "Numerical Study of a Staged Fuel Injection for Supersonic Combustion," *AIAA Journal,* Vol. 20, No. 10, Oct. 1982, pp. 1426-1427.

[152]Griffin, M. D., Anderson, J. D. Jr., and Diwaker, R., "Navier-Stokes Solutions of the Flowfield in an Internal Combustion Engine," *AIAA Journal,* Vol. 14, No. 12, Dec. 1976, pp. 1665-1666.

[153]Kothari, A. P., Anderson, J. D. Jr., and Jones, E., "Navier-Stokes Solutions for Chemical Laser Flows," *AIAA Journal,* Vol. 14, May 1976, pp. 702-703.

[154]Parthasarathy, K. N., Anderson, J. D. Jr., and Jones, E., "Downstream Mixing Gasdynamic Lasers—A Numerical Solution," *AIAA Journal,* Vol. 17, No. 11, Nov. 1979, pp. 1208-1215.

[155]Emanual, G., Cline, M. C., and Witte, K. H., "Laser Induced Disturbance with Application to a Low Reynolds Number Flow," *AIAA Journal,* Vol. 19, No. 2, Feb. 1981, pp. 226-231.

[156]Kee, R. J. and Miller, J. A., "A Split-Operator, Finite Difference Solution for Axisymmetric Laminar Jet Diffusion Hances," AIAA Paper 77-639, June 1977.

[157]Gough, P. S. and Zwarts, F. J., "Modeling Heterogeneous Two-Phase Reacting Flow," *AIAA Journal,* Vol. 17, No. 1, Jan. 1979, pp. 17-24.

[158]Markatos, N. C., Spalding, D. B., Tatchell, D. G., and Mace, A. C. H., "Flow and Combustion in the Base-Wall Region of a Rocket Exhaust Plume," *Combustion Science and Technology,* Vol. 28, 1982, pp. 15-29.

[159]Mulperu, S. R. and Wilkin, G. B., "Finite-Difference Calculations of Unsteady Premixed Flame-Flow Interactions," AIAA Paper 83-1917, July 1983.

[160]Brailovskaya, I. Y., "A Difference Scheme for Numerical Solution of the Two-Dimensional Nonstationary Navier-Stokes Equation for a Compressible Gas," *Soviet Physics-Doklady,* Vol. 10, No. 2, 1965, pp. 107-110.

[161]Kurzrosk, J. W. and Mates, R. E., "Exact Numerical Solutions of the Time-Dependent Compressible Navier-Stokes Equations," AIAA Paper 66-30, 1966.

[162]Thommen, H. U., "Numerical Integration of the Navier-Stokes Equations," *Zeitschrift für angewandte Mathematik und Physik,* Vol. 17, No. 3, 1966, pp. 369-384.

[163]Pavlov, B. M. "Calculation of Supersonic Flow Past Blunt Bodies Using the Complete Navier-Stokes Equations," *Fluid Dynamics Transactions,* Vol. 3, No. 3, 1968, pp. 88-90.

[164]Moretti, G. and Salas, M. D., "The Blunt Body Problem for a Viscous Rarefied Gas Flow," AIAA Paper 69-139, 1969.

[165]Allen, J. S. and Cheng, S. I., "Numerical Solutions of the Compressible Navier-Stokes Equations for the Laminar Near Wake," *Physics of Fluids,* Vol. 19, No. 1, 1970, pp. 37-52.

[166]Roache, P. J. and Mueller, T. J., "Numerical Solutions of Compressible and Incompressible Separated Flows," AIAA Paper 68-741, 1968.

[167]Brailovskaya, I. Y. "Calculation of Viscous Compressible Gas Flow Past a Corner," *Fluid Dynamics Transactions,* Vol. 2, No. 3, 1967, pp. 47-55.

[168]Carter, J. E., "Numerical Solutions of the Navier-Stokes Equations for the Supersonic Laminar Flow over a two-Dimensional Compression Corner," PhD Thesis, Virginia Polytechnic Institute and State University, Blacksburg, Va., 1971.

[169]Skoglund, V. J., Cole, J. K., and Staiano, E. F., "Numerical Techniques for Viscous Compressible Flows with Shock Waves," *Proceedings of the 1968 Heat Transfer and Fluid Mechanics Institute,* Stanford University Press, Stanford, Calif. pp. 151-173.

[170]MacCormack, R. W., "Numerical Solution of the Interaction of a Shock Wave with a Laminar Boundary Layer," *Proceedings of the Second International Conference on Numerical Methods in Fluid Dynamics,* Sept. 15-19, 1970, University of California, Berkeley, pp. 151-163.

[171]Cheng, S. I., "A Critical Review of the Numerical Solution of Navier-Stokes Equations," *Lecture Notes in Physics,* edited by H. J. Airz, Springer-Verlag, Berlin, 1974, pp. 78-225.

[172]MacCormack, R. W. and Lomax, H., "Numerical Solution of Compressible Viscous Flows,"*Annual Review of Fluid Mechanics,* Vol. 11, 1979, pp. 289-316.

[173]LeBalleur, J. C., Peyret, R., and Viviand, H., "Numerical Studies in High Reynolds Number Aerodynamic" *Computers and Fluids,* Vol. 8, No. 1, March 1980, pp. 1-30.

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

[175]MacCormack, R. W., "An Efficient Numerical Method for Solving the Time Dependent Compressible Navier-Stokes Equations at High Reynolds Number," *Proceedings of the ASME Winter Annual Meeting,* New York, 1976, pp. 49-64.

[176]MacCormack, R. W., "A Numerical Method for Solving the Equations of Compressible Viscous Flow," *AIAA Journal,* Vol. 20, No. 9, Sept. 1982, pp. 1275-1281.

[177]Napolitano, M., Werle, J. J., and Davis, R. T., "Numerical Technique for the Triple-Deck Problem," *AIAA Journal,* Vol. 17, No. 7, July 1979, pp. 699-705.

[178]Tannehill, J. C., Holst, T. L., and Raklich, J. V., "Numerical Computation of Two-Dimensional Viscous Blunt Body Flows with an Impinging Shock," *AIAA Journal,* Vol. 14, No. 2, Feb. 1976, pp. 204-211.

[179]Tannehill, J. C., Vigneron, Y. C., and Rakich, J. V., "Numerical Solution of the Two-Dimensional Turbulent Blunt Body Flows with an Impinging Shock," *AIAA Journal,* Vol. 17, No. 12, Dec. 1979, pp. 1289-1290.

[180]Lindemuth, I. and Killeen, J., "Alternating Direction Implicit Techniques for Two-Dimensional Magnetohydrodynamic Calculation," *Journal of Computational Physics,* Vol. 13, Oct. 1973, pp. 181-208.

[181]Briley, W. R. and McDonald, H., "Solution of the Three-Dimensional Compressible Navier-Stokes Equations by an Implicit Technique," *Proceedings of the Fourth International Conference on Numerical Methods in Fluid Dynamics,* Boulder, Colo., June 1974.

[182]Beam, R. and Warming, R. F., "An Implicit Factored Scheme for the Compressible Navier-Stokes Equations," *AIAA Journal,* Vol. 16, No. 4, April 1978, pp. 385-402.

[183]Holst, T. L., "Numerical Computation of Three-Dimensional Blunt Body Flow Fields with an Impinging Shock," PhD Dissertation, Iowa State University, Ames, Iowa, 1975.

[184]Kussoy, M. I., Viegas, J. R., and Horstman, C. C., "Investigation of a Three-Dimensional Shock Wave Separated Turbulent Boundary Layer," *AIAA Journal,* Vol. 18, No. 12, Dec. 1980, pp. 1477-1484.

[185]Brosh, A., Kussoy, M. I., and Hung, E. M., "Experimental and Numerical Investigation of the Impingement of an Oblique Shock Wave on a Body of Revolution," AIAA Paper 83-1757, Jan. 1983.

[186]Agarwal, R. K. and Bower, W. W., "Navier-Stokes Computations of Turbulent Compressible Two-Dimensional Impinging Jet Flowfields," *AIAA Journal,* Vol. 20, No. 5, May 1982, pp. 577-584.

[187]Jones, W. P. and Launder, B. E., "The Calculation of Low-Reynolds Number Phenomena with a Two-Equation Model of Turbulence," *International Journal of Heat and Mass Transfer,* Vol. 16, June 1973, pp. 1119-1129.

[188]Hung, C. M. and MacCormack, R. W., "Numerical Simulation of Supersonic and Hypersonic Turbulent Compression Corner Flows," *AIAA Journal,* Vol. 15, No. 3, March 1977, pp. 410-416.

[189]Horstman, C. C., Settles, G. S., Vas, I. E., Bogdonoff, S. M., and Hung, C. M., "Reynolds Number Effects on Shock-Wave Turbulent Boundary-Layer Interactions," *AIAA Journal,* Vol. 15, No. 8, Aug. 1977, pp. 1152-1158.

[190]Settles, G. S., Fitzpatrick, T. J., and Bogdonoff, S. M., "Detailed Study of Attached and Separated Compression Corner Flowfields in High Reynolds Number Supersonic Flow," *AIAA Journal,* Vol. 17, No. 6, June 1979, pp. 579-585.

[191]Viegas, J. R. and Horstman, C. C., " Comparison of Multiequation Turbulence Models for Several Shock Boundary Layer Interaction Flows," *AIAA Journal,* Vol. 17, No. 8, Aug. 1979, pp. 811-820.

[192]Horstman, C. C., Settles, G. S., Williams, D. R., and Bogdonoff, S. M., "A Reattaching Free Shear Layer in Compressible Turbulent Flow," *AIAA Journal,* Vol. 20, No. 1, Jan. 1982, pp. 79-85.

[193]Coakley, T. J. and Viegas, J. R., "Turbulence Modeling of Shock Separated Boundary-Layer Flows," Symposium on Turbulent Shear Flows, University Park, Pa., April 18-20, 1977.

[194]Coakley, T. J., Viegas, J. R., and Horstman, C. C., "Evaluation of Turbulence Models for Three-Primary Types of Shock Separated Boundary Layers," AIAA Paper 77-692, 1977.

[195]Visbal, M. R. and Knight, D. D., "Evaluation of the Baldwin-Lomax Turbulence Model for Two-Dimensional Shock Wave Boundary Equations," AIAA Paper 83-1697, 1983.

[196]Glushko, G. S., "Turbulent Boundary Layer on a Flat Plate in an Incompressible Fluid," *Bulletin of the Academy of Sciences of the USSR, Mechanics Series,* No. 4, 1965, pp. 13-23.

[197]Wilcox, D. C. and Rubesin, M. W., "Progress in Turbulence Modeling for Complex Flow Fields, Including Effects of Compressibility," NASA, TP-1517, April 1980.

[198]Suffman, P. G., "A Model for Inhomogeneous Turbulent Flow," *Proceedings of the Royal Society of London,* Vol. A317, 1970, pp. 417-433.

[199]Launder, B. E. and Spalding, D. B., *Mathematical Models of Turbulence,* Academic Press, London, 1972.

[200]Hung, C. M., "Numerical Solution of Supersonic Laminar Flow Over an Inclined Body of Revolution," *AIAA Journal,* Vol. 18, No. 8, Aug. 1980, pp. 921-928.

[201]Korkegi, R. H., "On the Structure of 3-D Shock Induced Separated Flow Regions," *AIAA Journal,* Vol. 14, No. 5, May 1976, pp. 697-700.

[202]Peake, D. J. and Tobak, M., "Three-Dimensional Interactions and Vortical Flows with Emphasis on High Speeds," AGARDHG No. 252, July 1980.

[203]Ghia, K. H. and Davis, R. T., "A Study of Compressible Potential and Asymptotic Viscous Flows for Corner Region," *AIAA Journal,* Vol. 12, March 1974, pp. 355-359.

[204]Mikhail, A. G. and Ghia, K. N., "Viscous Compressible Flow in the Boundary Region of an Axial Corner," *AIAA Journal,* Vol. 16, No. 9, Sept. 1978.

[205]Li, C. P., "A Numerical Study of the Axial Corner Flow," AIAA Paper 78-1146, July 1978.

[206]Shang, J. S. and Hankey, W. L., "Numerical Solution of the Navier-Stokes Equations for a Three-Dimensional Corner," *AIAA Journal,* Vol. 15, No. 11, Nov. 1977, pp. 1575-1582.

[207]Shang, J. S., Hankey, W. L., and Petty, J. S., "Three Dimensional Supersonic Interacting Turbulent Flow Along a Corner," *AIAA Journal,* Vol. 17, No. 7, July 1979, pp. 706-713.

[208]Hung, C. M. and MacCormack, R. W., "Numerical Solution of Supersonic Laminar Flow Over a 3-D Compression Corner," AIAA Paper 77-694, June 1977.

[209]Hung, C. M. and MacCormack, R. W., "Numerical Solution of Three-Dimensional Shock Wave and Turbulent Boundary-Layer Interaction," *AIAA Journal,* Vol. 16, No. 10, Oct. 1978, pp. 1090-1096.

[210]Horstman, C. C. and Hung, C. M., "Computation of Three-Dimensional Turbulent Separated Flows at Supersonic Speeds," *AIAA Journal,* Vol. 17, No. 11, Nov. 1979, pp. 1155-1156.

[211]Hung, C. M. and Kurasaki, S. S., "Thin-Layer Approximation for Three-Dimensional Supersonic Corner Flows," *AIAA Journal,* Vol. 18, No. 12, Dec. 1980, pp. 1544-1546.

[212]Osham, B., Vas, I. E., and Bogdonoff, S. M., "An Experimental Study of Three-Dimensional Flow Field in an Axial Corner at Mach 3," AIAA Paper 77-689, June 1977.

[213]Peake, D. T. and Rainbird, W. J., "The Three-Dimensional Separation of a Turbulent Boundary Layer by a Skewed Shockwave: and its Control by the Use of Tangential Air Injection," AGARD CP-168, Paper 40, May 1975.

[214]Kutler, P., Pulliam, T. H., and Vigneron, Y. C., "Computation of the Viscous Supersonic Flow over External Axial Corners," *AIAA Journal,* Vol. 17, No. 6, 1979, pp. 571-578.

[215]Dolling, D. S. and Bogdonoff, S. M., "Blunt Fin-Induced Shock Wave/Turbulent Boundary-Layer Interaction," *AIAA Journal,* Vol. 20, No. 12, Dec. 1982, pp. 1674-1680.

[216]Briley, W. R. and McDonald, H., "Computation of Three-Dimensional Horseshoe Vortex Flow Using the Navier-Stokes Equations," Seventh International Conference on Numerical Methods in Fluid Dynamics, Stanford, Calif., 1980.

[217]Tobak, M. and Peake, D. J., "Topology of Two-Dimensional and Three-Dimensional Separated Flows," AIAA Paper 79-1480, July 1979.

[218]Kline, S. J., Cantwell, B. J., and Lilley, G. M., eds., Conference on Complex Turbulent Flow, 1980-1981, AFOSR-HTTM-stand, Vols.1-3, Stanford University, Stanford, Calif., Vols. 1-3, 1982.

[219]Tenekes, H. and Lumley, J. L., *A First Course in Turbulence,* The MIT Press, Cambridge, Mass., 1972.

[220]Friedlander, S. K. and Topper, L., eds., *Turbulence: Classical Papers on Statistical Theory,* Interscience, New York, 1962.

[221]Roshko, A., "Structure of Turbulent Shear Flows: A New Look," *AIAA Journal,* Vol. 14, No. 10, Oct. 1976, pp. 1349-1357.

[222]Payne, R. B., "Calculation of Unsteady Viscous Flow Past a Circular Cylinder," *Journal of Fluid Mechanics,* Vol. 4, 1958, p. 81.

[223]Fromm, J. E., *Fundamental Methods in Hydrodynamics,* Academic Press, New York, 1964, p. 346.

[224]Scala, S. M. and Gordon, P., "Solution of the Time-Dependent Navier-Stokes Equations for the Flow Around a Circular Cylinder," *AIAA Journal,* Vol. 6, No. 5, 1968, pp. 815-822.

[225]Jordan, S. K. and Fromm, J. E., "Oscillatory Drag, Lift, and Torque on a Circular Cylinder in a Uniform Flow," *Physics of Fluids,* Vol. 15, No. 3, March 1972, pp. 371-376.

[226]Thoman, D. C. and Szewczyk, A. A., "Time-Dependent Viscous Flow Over a Circular Cylinder," *Physics of Fluids,* Supplement II, 1969, pp. 1176-1186.

[227]Shang, J. S., "Oscillatory Compressible Flow Around a Cylinder," AIAA Paper 82-0098, 1982.

[228]Kumar, A. and Salas, M. D., "Euler and Navier-Stokes Solutions for Supersonic Shear Flow Past a Circular Cylinder," AIAA Paper 84-0339, Jan. 1983.

[229]Kawanura, T. and Kuwahara, K., "Computation of High Reynolds Number Flow Around a Circular Cylinder with Surface Roughness," AIAA Paper 84-0340, 1984.

[230]McCroskey, W. J. and Pucci, L., "Viscous-Inviscid Interaction on Oscillating Airfoils in Subsonic Flow," *AIAA Journal,* Vol. 20, No. 2, Feb. 1982, pp. 167-174.

[231]Rockwell, D. and Naudascher, E., "Self-Sustained Oscillations of Impinging Free Shear Layers," *Annual Review of Fluid Mechanics,* Vol. 11, 1979, pp. 67-94.

[232]Hankey, W. L. and Shang, J. S., "Analysis of Self-Excited Oscillations in Fluid Flows," AIAA Paper 80-1346, July 1980.

[233]Rockwell, D., "Oscillation of Impinging Shear Layers," AIAA Paper 82-0047, Jan. 1982.

[234]Hankey, W. L. and Shang, J. W., "Analyses of Pressure Oscillations in an Open Cavity," *AIAA Journal,* Vol. 18, No. 8, Aug. 1980, pp. 892-898.

[235]Widhopf, G. F. and Victoria, K. J., "Numerical Solution of the Unsteady Navier-Stokes Equations for the Oscillatory Flow Over a Concurve Body," *Lecture Notes in Physics,* No. 35, Springer-Verlag, New York, June 1974, pp. 431-444.

[236]Rakich, J. V., Vigneron, Y. E., and Tannehill, J. C., "Navier-Stokes Calculations for Laminar and Turbulent Supersonic Flow Over Indented Nosetips," AIAA Paper 78-260, Jan. 1978.

[237]Kutler, P., Chakravarthy, S. R., and Lombard, C. K., "Supersonic Flow Over Ablated Nosetips Using Implicit Numerical Procedure," AIAA Paper 78-213, Jan. 1978.

[238]Shang, J. S., Hankey, W. L., and Smith, R. E., "Flow Oscillations of Spike-Tipped Bodies," *AIAA Journal,* Vol. 20, No. 1, Jan. 1982, pp. 25-26.

[239]Calarese, W., Hankey, W. L. Jr., and Shang, J. S., "Investigation of Self-Sustained Shock Oscillations on a Spike-Tipped Body at Mach 3," *Computers in Flow Predictions and Fluid Dynamics Experiments,* ASME Winter Annual Meeting, Nov. 15-20, 1981, pp. 151-156.

[240]Newsome, R. W. Jr., "Numerical Simulation of Near Critical and Unsteady Subcritical Inlet Flow Fields," AIAA Paper 83-0175, Jan. 1983.

[241]Liou, M. S. and Coakley, T. J., "Numerical Simulations of Unsteady Transonic Flow in Diffusers," AIAA Paper 82-1000, June 1982.

[242]Scott, J. N. and Hankey, W. L. Jr., "Numerical Simulation of Cold Flow in an Axisymmetric Centerbody Combustor," AIAA Paper 83-1741, July 1983.

[243]Purohit, S. C., Shang, J. S., and Hankey, W. L. Jr., "Numerical Simulation of Flow Around a Three-Dimensional Turret," *AIAA Journal,* Vol. 21, No. 11, Nov. 1983, pp. 1533-1540.

[244]Purohit, S. C., Shang, J. S., and Hankey, W. L. Jr., "Effect of Suction of the Wake Structure of a Three-Dimensional Turret," AIAA Paper 83-1738, July 1983.

[245]Fasel, H., "Investigation of the Instability of Boundary Layers by a Finite-Difference Model of the Navier-Stokes Equations," *Journal of Fluid Mechanics,* Vol. 78, No. 2, 1976, pp. 355-383.

[246]Orszag, S. A. and Patena, A. T., "Secondary Instability of Wall-Bounded Shear Flows," *Journal of Fluid Mechanics,* Vol. 128, 1983, pp. 347-385.

[247]Hankey, W. L. Jr. and Shang, J. S., "Natural Transition—A Self-Excited Oscillation," AIAA Paper 82-1011, June 1982.

[248]Leonard, A., "Large Eddy Simulation Techniques," AIAA Paper 83-1878, July 1983.

[249]Ashurst, W. T., "Large Eddy Simulation via Vortex Dynamics,"AIAA Paper 83-1879, July 1983.

[250]Morchoisne, Y., "Large Eddy Simulation Techniques," AIAA Paper 83-1880, July 1983.

[251]Shang, J. S., "Implicit-Explicit Method for Solving the Navier-Stokes Equations," *AIAA Journal*, Vol. 16, No. 5, May 1978, pp. 491-502.

[252]Rizk, Y. M. and Chaussee, D. S., "Three-Dimensional Viscous Flow Computations Using a Directionally Hybrid Implicit-Explicit Procedure," AIAA Paper 83-1910, July 1983.

[253]Hasse, W., Wagner, B., and Jameson, A., "Development of a Navier-Stokes Method Based on a Finite Volume Technique for the Unsteady Euler Equations," 5th GAMM Conference for Numerical Methods in Fluid Dynamics, Rome, Oct. 1983.

[254]Von Lavante, E. and Thompkins, W. T. Jr., "An Implicit, Bidiagonal Numerical Method for Solving the Navier-Stokes Equations," *AIAA Journal,* Vol. 21, No. 6, June 1983, pp. 828-833.

[255]Greenberg, J. B., "Semi-Implicit Hopscotch-Type Methods for the Time Dependent Navier-Stokes Equations," *AIAA Journal,* Vol. 20, No. 8, Aug. 1982, pp. 1064-1070.

[256]Khosla, P. K. and Rubin, S. G., "A Composite Velocity Procedure for the Compressible Navier-Stokes Equations," *AIAA Journal,* Vol. 21, No. 11, Nov. 1983, pp. 1546-1551.

[257]Brandt, A., "Multi-Level Adaptive Technique (MALT) for Fast Numerical Solution to Boundary Value Problem," *Lecture Notes in Physics,* Vol. 18, Springer-Verlag, Berlin, 1973, pp. 82-89.

[258]Orszag, S. A., "Spectral Methods of Problems in Complex Geometries," *Numerical Method of PDE's,* Academic Press, New York, 1979.

[259]Gottlieb, D. and Orszag, S. A., "Numerical Analysis of Spectral Methods: Theory and Applications," *SIAM,* Vol. 26, 1977.

[260]Lomax, H., "Some Prospects for the Future of Computational Fluid Dynamics," *AIAA Journal,* Vol. 20, No. 8, Aug. 1982, pp. 1033-1043.

[261]Chima, R. V. and Johnson, G. M., "Efficient Solution of the Euler and Navier-Stokes Equations with a Vectorized Multiple-Grid Algorithm," AIAA Paper 83-1893, July 1983.

[262]Davis, R. L., "The Prediction of Compressible Laminar Viscous Flows Using a Time-Marching Control Volume and Multigrid Technique," AIAA Paper 83-1896, 1983.

[263]Plotkin, A., "Spectral Method Solution for Some Laminar Channel Flows with Separation," *AIAA Journal,* Vol. 20, No. 12, Dec. 1982, pp. 1713-1719.

[264]Plotkin, A., "Development of Confirmed Laminar Wakes at Large Reynolds Number," *AIAA Journal,* Vol. 20, Feb. 1982, pp. 211-217.

[265]Kumar, A. and Yajnik, K. S., "Internal Separated Flows at Large Reynolds Number," *Journal of Fluid Mechanics,* Vol. 97, Pt. 1, 1980, pp. 27-51.

[266]Hussaini, M. Y., Baldwin, B. S., and MacCormack, R. W., "Asymptotic Features of Shock-Wave Boundary-Layer Interaction," *AIAA Journal,* Vol. 18, No. 8, Aug. 1980, pp. 1014-1016.

[267]Reddy, K. C., "Pseudo Spectral Approximation in a Three-Dimensional Navier-Stokes Code," *AIAA Journal,* Vol. 21, No. 8, Aug. 1983, pp. 1208-1210.

[268]Turkel, E., "On the Practical Use of Higher-Order Methods for Hyperbolic Systems," *Journal of Computational Physics,* 35, 1980, pp. 319-340.

[269]Rubin, S. G. and Khosla, P. K., "Higher-Order Numerical Solution Using Cubic Splines," *AIAA Journal,* Vol. 14, No. 7, July 1976, pp. 851-858.

[270]McCartin, B. J., "Applications of Exponential Splines in Computational Fluid Dynamics," *AIAA Journal,* Vol. 21, No. 8, Aug. 1983, pp. 1059-1065.

[271]Dwyer, H. A., Kee, R. J., and Sanders, B. R., "Adaptive Grid Method for Problems in Fluid Mechanics and Heat Transfer," *AIAA Journal,* Vol. 18, No. 10, Oct. 1980, pp. 1205-1212.

[272]Rai, M. M. and Anderson, D. A., "Application of Adaptive Grids to Fluid-Flow Problems with Asymptotic Solutions," *AIAA Journal,* Vol. 20, No. 4, April 1982, pp. 496-502.

[273]Shang, J. S., Buning, P. G., Hankey, W. L., and Wirth, M. C., "Performance of a Vectorized Three-Dimensional Navier-Stokes Code on the CRAY-1 Computer," *AIAA Journal,* Vol. 18, No. 9, Sept. 1980, pp. 1073-1079.

[274]Smith, R. E. and Pitts, J., "The Solution of the Three-Dimensional Viscous Compressible Navier-Stokes Equations on a Vector Computer," *Proceedings of the Third IMACS Symposium,* 1979, pp. 245-252.

[275]Benek, J. A., Steger, J. L., and Dougherty, F. C., "A Flexible Grid Embedding Technique with Application to the Euler Equations," AIAA 83-1944, July 1983.

[276]Teman, R., "Navier-Stokes Equations," North-Holland, Amsterdam, 1979.

[277]Garabedian, P. R., *Partial Differential Equations,* John Wiley, New York 1964, p. 70.

[278]Steger, J. L., Pulliam, T. H., and Chima, R. V., "An Implicit Finite Difference Code for Inviscid and Viscous Cascade Flow," AIAA Paper 80-1427, July 1980.

[279]Rudy, D. H. and Strikwerda, J. C., "Boundary Conditions for Subsonic Compressible Navier-Stokes Calculations," *Computers and Fluids*, Vol. 9, 1981, pp. 327-338.

[280]Bradshaw, P., Cebeci, T., and Whitelaw, J., *Engineering Calculation Methods for Turbulent Flow,* Academic Press, New York, 1981.

[281]Reynolds, W. C., "Computation of Turbulent Flows," *Annual Reveiw of Fluid Mechanics,* Vol. 8, 1976, pp. 183-208.

[282]Marvin, J. G., "Turbulence Modeling for Computational Aerodynamics," *AIAA Journal,* Vol. 21, No. 7, July 1983, pp. 941-955.

[283]Cebeci, T., "Calculation of Compressible Turbulent Boundary Layers with Heat and Mass Transfer," *AIAA Journal,* Vol. 9, No. 6, June 1971, pp. 1091-1097.

[284]Lomax, H. and Pulliam, T. H., "A Fully Implicit Factored Code for Computing Three-Dimensional Flows on the ILLIAC IV," *Parallel Computations,* edited by G. Rodrique, Academic Press, New York, 1982, pp. 217-250.

[285]Thompson, J. F., "A Survey of Grid Generation Techniques in Computational Fluid Dynamics," AIAA Paper 83-0447, Jan. 1983.

[286]Ives, D. C., "Quasi Three-Dimensional Grid Generation Using Conformal Mapping," AIAA Paper 83-1906, July 1983.

[287]Thompson, J. F. and Warsi, Z. U. A., "Three-Dimensional Grid Generation from Elliptic Systems," AIAA Paper 83-1905, July 1983.

[288]Steger, J. L. and Chaussee, D. S., "Generation of Body-Fitted Coordinates Using Hyperbolic Partial Differential Equations," *SIAM Journal of Scientific Statistics Computations*, 1, 1980, pp. 431-437.

[289]Eiseman, P. R., "A Multisurface Method of Coordinate Generation," *Journal of Computational Physics,* Vol. 33, No. 1, Oct. 1979, pp. 118-150.

[290]Smith, R. E., "Three-Dimensional Algebraic Grid Generation," AIAA Paper 83-1904, July 1983.

[291]Belie, R. G. and Rapagnani, N. L., "Color Computer Graphics," *Astronautics and Aeronautics,* June 1981, pp. 4-48.

[292]Gregory, T. J. and Carmichael, R. L., "Getting the Picture Through Computer Graphics," *Astronautics and Aeronautics,* April 1983, pp. 16-29.

[293]Korkegi, R. H., "The Impact of CFD on Development Test Facilities—A National Research Council Projection," AIAA Paper 83-1764, 1983.

[294]National Research Council, *Influence of Computational Fluid Dynamic Upon Experimental Aerospace Facilities: A Fifteen Year Projection,* National Academy Press, Washington, D. C., 1983.

[295]Peak, D. J. and Tobak, M., "On Issues Concerning Flow Separation and Vortical Flows in Three-Dimensions," NASA TM 84374, June 1983.

[296]Cornish, J. J. III, "Vortex Flows," AIAA Paper 83-1812, July 1983.

[297]Kerrebrock, J. L., "Flow in Transonic Compressors," *AIAA Journal,* Vol. 19, No. 1, Jan. 1981, pp. 4-19.