Lecture Notes in Physics

323

D.L. Dwoyer M.Y. Hussaini
R.G. Voigt (Eds.)

11th International Conference on Numerical Methods in Fluid Dynamics

Springer-Verlag
Berlin Heidelberg New York Tokyo

Editors

D. L. Dwoyer
M. Y. Hussaini
R. G. Voigt
ICASE, NASA Langley Research Center
Hampton, VA 23665-5335, USA

ISBN 3-540-51048-6 Springer-Verlag Berlin Heidelberg New York
ISBN 0-387-51048-6 Springer-Verlag New York Berlin Heidelberg

Printing: Druckhaus Beltz, Hemsbach/Bergstr.
Binding: J. Schäffer GmbH & Co. KG., Grünstadt
2158/3140-543210 – Printed on acid-free paper

DEDICATION

This symposium is dedicated to Professor Maurice Holt on the occasion of his seventieth birthday in tribute to his four decades of research in applied mathematics and fluid mechanics. Since he is still very active in research and considers the last four decades as but the first phase of his research career, a final evaluation of his contributions is premature. However, it is only proper that his biography be briefly sketched with a mention of honors and recognitions he has already won.

Professor Maurice Holt was born on May 16, 1918 in Wildboarclough, Cheshire, England. He was educated at the Manchester Grammar School and the University of Manchester in England. He did his Masters thesis with T. G. Cowling in 1944, and his doctorate with Sydney Goldstein in 1948. Immediately after graduation, he joined the University of Liverpool as a lecturer in Mathematics, and a year later moved to the University of Sheffield in England. In 1952, he joined the Ministry of Supply where he served as the Principal Scientific Officer in charge of the Theoretical Aerodynamics Section of the Applied Mathematics Division, Armament Research and Development Establishment, Fort Halstead, Kent. In 1955, he was a visiting lecturer in the Mathematics Department at Harvard University at the invitation of Garrett Birkhoff. In 1956, he entered the United States, and joined the faculty of the Division of Applied Mathematics at Brown University. Since 1960, he has been Professor of Aeronautical Sciences at the University of California at Berkeley.

Professor Maurice Holt's scientific work is too extensive to be discussed in detail here. Suffice it to say, then, that his research in fluid mechanics has encompassed a diverse range of physical problems. Supersonic and transonic aerodynamics, blast waves, underwater explosions and supersonic separated flows are some of the subjects on which his work has received international recognition. He is one of the pioneers in the field of computational fluid dynamics. Not only did he develop some numerical techniques that were widely used during the 1960's for supersonic blunt body problems, conical flows and separated flows, but he also is a leading figure in the establishment of East-West cooperation in computational fluid dynamics. In 1969, he and Academician Belotserkovskii together started the highly successful conference series, the International Conference on Numerical Methods in Fluid Dynamics.

As an excellent applied mathematician, Professor Holt's keen insight into the physics of fluids has become a trademark of his work. His work has been fundamental and applied, in the best scientific tradition. He has been a valuable consultant to both industry and government at various times. Lockheed Aircraft Corporation, Northrop Corporate Laboratories, The Aerospace Corporation, and Lawrence Berkeley Laboratory are some of the organizations which have profited from his expertise. He has been a visiting professor at a number of international institutions such as Université Paris, and Université Pierre et Marie Curie. As an excellent teacher, he has taught and trained nearly three dozen students who have themselves become leaders in research. He has served as editor of several journals, including the ASME Journal of Applied Mechanics. In recognition of his pioneering research contributions to fluid mechanics and his training of two generations of students, he has been elected a fellow of the American Society of Mechanical Engineers and the American Physical Society. His recognition extends beyond the western world; he was invited to be the guest of the USSR Academy of Sciences and the Romanian Academy of Sciences because of his leadership in fluid mechanics research.

It is with affection for the deep impact that he has had on our lives and with respect for his accomplishments as a teacher and a scholar that we dedicate this symposium to Professor Maurice Holt. We only hope that our interaction with him will continue for many years to come.

W. F. Ballhaus, Jr.
M. Y. Hussaini

Editors' Preface

The present volume of Lecture Notes in Physics covers the proceedings of the Eleventh International Conference on Numerical Methods in Fluid Dynamics, held in Williamsburg, Virginia, June 27-July 1, 1988. It contains 103 papers. Seven of these papers are based on invited lectures, and the rest were selected on the basis of abstracts submitted from all over the world by four paper selection groups, one in the U.S.A. headed by Maurice Holt, another in Europe headed by Roger Temam, the third in U.S.S.R. headed by Victor Rusanov, and the fourth in Japan (representing the Pacific Rim Countries) headed by K. Oshima. Following the usual tradition, the invited papers appear first, and then the contributed papers in alphabetical order by first author.

The conference co-chairmen were D. L. Dwoyer and Robert G. Voigt; they are indebted to many who helped with the detailed organization of the meeting; they thank all of them, and in particular, Ms. Mary Adams who was in charge of the computer graphics displays and Ms. Emily Todd, the conference secretary.

The financial support was provided by NASA Langley Research Center.

We thank Dr. W. Beiglböck and the editorial staff of Springer-Verlag for assistance in preparing the proceedings.

November 1988.

D. L. DWOYER M. Y. HUSSAINI R. G. VOIGT
(Editors)

CONTENTS

Computational Fluid Dynamics – A Personal View

M.Y. Hussaini
Institute for Computer Applications in Science and Engineering
NASA Langley Research Center
Hampton, VA 23665

I. Abstract

This paper provides a personal view of computational fluid dynamics. The main theme is divided into two categories – one dealing with algorithms and engineering applications and the other with scientific investigations. The former category may be termed computational aerodynamics with the objective of providing reliable aerodynamic or engineering predictions. The latter category is essentially basic research where the algorithmic tools are used to unravel and elucidate fluid dynamic phenomena hard to obtain in a laboratory. While dealing with the algorithms, only the underlying principles are discussed and the engineering applications are omitted. A personal critique of the numerical solution techniques for both compressible and incompressible flows is included. The discussion on scientific investigations deals in particular with transition and turbulence. In conclusion, some challenges to computational fluid dynamics are mentioned, the grand challenge being turbulence and reacting flows.

II. Introduction

Computational fluid dynamics can broadly be divided into three categories – 1) algorithms, 2) scientific investigations, and 3) engineering applications. Algorithms may be considered as a common foundation for engineering applications and scientific investigations. Engineering applications include many fields such as aerodynamics, meteorology, astrophysics and oceanography. Scientific investigations involve unravelling or elucidating fundamental flow phenomena hard to obtain in a laboratory. In this paper, the principles underlying the algorithms are briefly discussed; next some current engineering applications in aerodynamics are presented; and finally, some representative results of scientific investigations are described in the area of my interest which is stability and transition to turbulence.

The discussion of the algorithms will distinguish between compressible and incompressible flows. For most purposes, the Navier-Stokes equations provide the best description of compressible flows excluding the rarefied gas dynamics regime. These equations do not require further elaboration. From the point of view of numerical discretization of these continuum equations, the inviscid terms have provided the greatest challenge. The algorithms for the inviscid equations, which are usually called the compressible Euler equations, can be extended in a technically straightforward fashion to include viscous terms. Usually, some equivalent of central differencing is employed to discretize the viscous terms. For this reason, this discussion of algorithms for compressible flows will henceforth focus on the Euler equations. They must be viewed, not in isolation, but rather as a limit of the Navier-Stokes equations as the Reynolds number tends to infinity. The physically relevant solutions are those which satisfy additional constraints such as the entropy condition.

The most distinguishing feature of compressible flows is the presence of shocks. Any compressible flow algorithm must be capable of handling a shock wave which means that the Rankine-Hugoniot conditions must be satisfied across the shock. A sufficient condition for this is that the equations be in the conservation form and the numerical scheme be both conservative and dissipative. A shock wave captured in a discrete solution tends to be somewhat diffused, and the solution may have attendant oscillations depending on the details of the scheme and the dimensionality of the problem. Most practitioners rate compressible flow algorithms according to the cosmetic appearance of their solutions to standard test problems; the criteria are how sharp the captured shock is and how well the wiggles are suppressed. A discrete shock will always remain a fertile ground of research for the numerical analysts.

Compressible flow algorithms should also be able to preclude expansion shocks. For instance, the Lax-Wendroff [17] scheme does admit expansion shocks; a remedy for this was provided by Majda and Osher [20] almost a decade later using the entropy condition.

A contact discontinuity poses a different problem for discrete solutions. Although one does not have to satisfy the entropy condition across the discontinuity, one has to deal with the problem of the physical instability of such vortex sheets. The numerical treatments that I am aware of suppress such instabilities through artificial viscosity or compressibility. Again, algorithms are rated according to the sharpness of the contact discontinuity and the smoothness (absence of wiggles) of the solution across the discontinuity. Contact discontinuities are not as ubiquitous as shocks, and seem to have attracted less attention than shocks.

Real gas effects occur in the hypersonic regime and require additional reaction-diffusion equations for the species to be solved. This may add to the stiffness of the problem.

The most obvious examples of singularities are: the intersection of a shock wave with a surface, the triple shock point, and the core of a rolled-up vortex. Discrete Euler solutions in such situations may yield, owing to inherent diffusion, results that tend to agree with experimental observation. The purist will always be uneasy about such agreement.

Let me discuss briefly the principles underlying compressible flow algorithms. The Euler equations are but one example of a hyperbolic system of conservation laws. The fundamental physical principle is the conservation law in the integral form

$$\frac{d}{dt}\int \vec{U}\cdot d\vec{r} = -\int \vec{F}\cdot\hat{n}\, dA. \tag{1}$$

Subject to the uniform continuity of $\vec{U}$ and the differentiability of $\vec{F}$, this reduces to the partial differential equation

$$\vec{U}_t = -\nabla\cdot\vec{F}. \tag{2}$$

Consider now the one-dimensional scalar case and integrate the conservation law across a single cell. This yields the semi-discrete form

$$\frac{\partial \bar{U}_j}{\partial t} = \frac{F_{j+1/2} - F_{j-1/2}}{\Delta x} \tag{3}$$

where the cell average quantity is

$$\bar{U}_j = \int_{x_{j-1/2}}^{x_{j+1/2}} U_j(\xi)\, d\xi \tag{4}$$

and the interface flux is

$$F_{j+1/2} = F(U_{j+1/2}). \tag{5}$$

Note the distinction between pointwise and average values. A historical perspective is useful for appreciating modern techniques for solving Equation 1, which is the first building block of compressible flow algorithms.

III. Compressible Flow Algorithms

A. *The Sixties*

The two representative methods of the 1960's are Godunov [8] and Lax-Wendroff (L-W) [17] schemes.

Godunov's method exploits the local physics of the hyperbolic problem, i.e., the local characteristics of the problem. It solves the Riemann problem

$$W_t + F(W)_x = 0 \tag{6}$$

where

$$\begin{aligned} W(x; U_L; U_R) &= U_L, \quad x < 0 \\ &= U_R, \quad x > 0 \end{aligned} \tag{7}$$

with piecewise constant initial conditions, and uses this solution to compute the flux function at time level n

$$F^n_{j+1/2} = F(W(0; U^n_j, U^n_{j+1}). \tag{8}$$

It is first-order accurate, has sufficient numerical viscosity for stability, and virtually always works in the sense that it is rarely unstable. The Lax-Wendroff method, on the other hand, is based on Taylor series expansion, and is second order accurate. The two-step version of the method,

$$U^*_{j+1/2} = \frac{1}{2}(U^n_{j+1} + U^n_j) - \frac{\Delta t}{\Delta x}(F^n_{n+1} - F^n_j) \tag{9}$$

$$U^{n+1}_j = U^n_j - \frac{\Delta t}{\Delta x}(F^*_{j+1/2} - F^*_{j-1/2}) \tag{10}$$

is due to Richtmyer [28]. It contains a mild amount of implicit dissipation; additional artificial damping must be explicitly added to control oscillations.

The advantages and disadvantages of these two approaches are apparent in their respective solutions to the now standard one-dimensional shock tube test problem. Shortly after the flow is initiated, there is a shock wave, rarefaction wave, and a contact discontinuity. These features are evident in fig. 1 where the exact solution is denoted by a solid line. In comparison with the Godunov's method, the L-W scheme furnishes a sharper shock at the expense of greater dispersion. The L-W results here purposely did not utilize artificial viscosity (to highlight the dispersion errors); in practice explicit damping is always present, but with a magnitude at the discretion of the user.

These two schemes may now be out-dated, but the basic principles and dilemmas of the methods persist to this day.

B. *The Seventies*

In the 1970's numerous sequels appeared, some of which are discussed below.

MacCormack's scheme [19], an efficient and robust variant of the Lax-Wendroff scheme has been a workhorse of industrial applications. It is the classic example of a good scheme which comes along at the right time, and soon proves its worth on some non-trivial problems of its day.

Flux-Corrected-Transport (FCT) [1] may be considered as the first example of the total variation diminishing schemes (TVD), although the TVD concept itself was introduced a decade later [12]. It consists of two steps; in the first step, a low-order solution is obtained, and is used to determine that fraction of the antidiffusion flux that can be applied at the second step to produce, in some sense, maximum steepening of the solution, while precluding any new extrema. It is a modular, scalar approach suitable for a single conservation law. For systems, some secondary effects of flux limiters are found, and there appears to be room for improvement.

The hybrid scheme, originally due to Harten [9] is a high order scheme away from the shocks and first order monotonic at the shock.

The random choice method [6] is in effect Godunov's method where random sampling of the Riemann solution replaces cell-averages. Although not conservative, the absence of averaging errors makes it the most accurate scheme. Unfortunately, of all the algorithms listed here, it suffers the most deterioration when extended beyond one dimension.

MUSCL [34,35] is a second order sequel to Godunov's method where the piecewise constant initial conditions for the Riemann problem are replaced by piecewise linear initial conditions with proper switching functions to avoid any new extrema. It may appear surprising that nearly two decades elapsed before an improvement to the Godunov's method was developed. However, despite its ingenuity, it was largely ignored, only in part due to the unpleasantness of coding up a general Riemann solver, and to the uncertainty of extending it beyond one dimension. The real hurdle proved to be the difficulty of devising robust flux limiters for the reconstruction phase.

The Λ−scheme [21] is based on the physics of signal propagation for the hyperbolic laws. It casts the Euler equations in the form of compatibility equations and uses a finite-difference discretization which conforms to the nature of the local characteristic directions. Incorporation of boundary conditions is efficient.

The artificial compression method (ACM) [10,11], in contrast to the artificial viscosity methods (AVM) [22], sharpens the contact discontinuities and shocks. It is the best method that I know of for contact discontinuities.

C. The Eighties

The 1980's have witnessed great advances in numerical techniques for compressible flows. It is still premature to select the methods which will prove to be the most enduring. The following list presents the leading contenders as well as a few long shots.

Flux splitting [31] for the Euler equations may be considered as characteristic differencing in conservation form. The numerical flux function is split into a sum of forward flux and backward flux, and this facilitates upwind differencing. Sometimes the flux differences are split rather than the flux themselves, and some improvements are achieved thereby [29].

The focus in the 1980's has been on high-order methods. High-order FCT [39], piecewise parabolic methods (PPM) [38], essentially non-oscillatory schemes (ENO) [13,14], and spectral methods were developed for hyperbolic problems. PPM and ENO have been applied to some complicated two-dimensional shocked flows. Spectral methods are relatively in the initial stages of development. Multigrid procedures [23] and TVD concepts [3] have been put to use in practical three dimensional applications in many laboratories and in some industries.

Figure 2 demonstrates how well a 1980 technique such as Phil Roe's flux-difference algorithm performs in the case of the standard shock tube test problem. The shock and contact discontinuities captured by this method are relatively sharp.

Total pressure and surface streamlines from the numerical simulation of a flow-field about an F-18 forebody for an incidence angle of 30° and a Reynolds of 740,000 is illustrated in figure 3. In the first phase of the simulation, after intrinsic consistency checks are performed, one examines how well the gross features of the flow-field are captured. In this case, the primary separation and secondary separation on the forebody, leading edge separation and the secondary separation on the strake are important features. These features are found to be in remarkable agreement with oil-flow visualization pictures from the experiment, not shown here.

Further engineering applications can be found in the paper by Ajay Kumar in these proceedings.

Compressible flow algorithms run into difficulties or otherwise become extremely inefficient at very low Mach numbers due to the fact that the equations become time-singular. In the near-zero Mach number limit, most CFD calculations are performed with algorithms designed specifically for incompressible flow. The history of these methods is indeed quite different from that of compressible flows methods.

IV. Incompressible Navier-Stokes Equations

The basic equations for incompressible viscous flow are well-known. The viscous terms introduce an intrinsic algorithmic difficulty in contrast to the compressible case. The divergence-free constraint, and in practice the viscous terms, must be treated implicitly; the simultaneous enforcement of these conditions is quite challenging. The boundary conditions involve only the velocity – there is no boundary condition on the pressure. Although it is common practice to derive a Neumann boundary condition on the pressure from the momentum equation, this is not always a fool-proof approach. The initial conditions are not as innocuous as they appear, for if they are not divergence-free, then the solutions are apt to have a time singularity.

Many algorithms treat the nonlinear advection terms explicitly and the linear terms implicitly. In this case, the most time-consuming part of the algorithm is the solution of the discrete Stokes problem

$$\begin{pmatrix} I+V & G \\ G^* & 0 \end{pmatrix} \begin{pmatrix} \vec{U} \\ p \end{pmatrix} = (A) \begin{pmatrix} \vec{U} \\ p \end{pmatrix} = \begin{pmatrix} \vec{f} \\ 0 \end{pmatrix} \tag{11}$$

where I is the identity operator, V the discrete viscous operator, G the discrete gradient operator (and G^* is its adjoint, which is the negative of the divergence operator). The essential difficulty is that A is too large for a direct solution, (remember we are interested in three-dimensional solutions) and its indefiniteness poses severe challenges for conventional iterative methods.

The major efforts on high-speed flow algorithms have focussed on improving the basic discretization. On the other hand, the most significant milestones for incompressible flows have been those which have improved the efficiency of solving the Stokes problem. A few of these are discussed below, and let me forthwith apologize for neglecting such alternatives as discrete vortex methods with random-walk modelling of diffusion.

The marker-and-cell method [15] was the first consistent numerical method for solving the incompressible Navier-Stokes equations with neither spurious modes, nor artificial pressure boundary condition. It did, however, suffer from the limitation that the viscous terms were treated explicitly.

The artificial compressibility method [4] adds to the continuity equation a fictitious term involving the product of a small parameter and the time-derivative of pressure. Thus it removes indefiniteness but sacrifices incompressibility. It is a method for steady-state solution.

The operator splitting methods [5,33], or the projection methods as they are sometimes called, uncouples the advection-diffusion terms from the incompressibility constraint. They involve two steps. In the first step, advection-diffusion terms are advanced, and in the second step, the pressure correction is carried out to enforce a divergence-free condition. Intermediate boundary conditions are important for such methods. They became available in 1971 [7].

Multigrid procedures have been used to obtain grid-independent convergence rates in the case of both finite-difference and spectral discretizations of the Navier-Stokes equations [2].

Although spectral methods have a long history, they were first demonstrated to be practical for high resolution calculations of incompressible flows in 1971 [25]. This, however, was purely for the periodic case where such difficulties as spurious modes and pressure boundary conditions are not present. Moreover, in this special case, the discrete divergence and viscous operators commute, a property which is lost for nonperiodic problems. It was not until 1978 that these difficulties were overcome for wall-bounded flows [26].

By virtue of certain advantages of spectral methods, such as exponential convergence, minimal phase errors, etc., they have become a dominant tool in simulations of instability and transition to turbulence and turbulent shear flows.

Now, I propose to present one example of transition simulation. A direct simulation of laminar breakdown in a flat plate boundary layer was performed and the data postprocessed for qualitative comparison against experimental results. Bubbles (in the experiment) and particles (in the computation) are periodically released from a wire normal to the plate in the vicinity of the peak plane (where the upwash is maximum). The results are shown in fig. 4.

V. Assessment of CFD Status

The following chart (fig. 5) lends some perspective. The horizontal axis measures the flow complexity and the vertical axis the geometric complexity. The ultimate goal of computational aerodynamics lies in the upper right hand corner, and is far beyond any forseeable capability. The Euler solutions to three dimensional flow past wings takes about 5 to 10 minutes on the Cray-2. The Reynolds-averaged Navier-Stokes solutions take about 8 to 10 hours; however, these algorithms have been made efficient through multigrid procedures, and this calculation time now ranges between 1 to 2 hours [30]. The increased computer resources taken by the Reynolds averaged Navier-Stokes calculations are due in part to the increased complexity of the equations themselves and in part to the additional spatial scales that must be resolved. In the direct simulations the range of spatial scales is orders of magnitude larger, and in addition, it is necessary to resolve an equally large range of temporal scales. Hence an order of magnitude increase in computer time is required.

Since we are so far removed from our ultimate goal, a key question is: should we be encouraged or discouraged by our past achievements, our current rate of progress, and our future prospects. Certainly, judging by some of the more extravagant claims that have been made in recent years, discouragement is in order. We should remember however that CFD is barely three decades old, whereas the related experimental and theoretical disciplines have flourished for more than a century. To borrow a phrase from Peter Lax [18], CFD is just "beginning to emerge, like Hercules from his cradle".

A cursory look through the "Current Capabilities and Future Directions in Computational Fluid Dynamics", the findings and recommendations of a NRC committee [24], reveals the following: 1) CFD has become an integral part of the design cycle in the aircraft industry; 2) solution algorithms for the Euler equations are reaching maturity for practical flow configurations, but those for the three-dimensional Navier-Stokes equations are at a relatively primitive stage; 3) in hypersonics, CFD is expected to be the principal source of flow information for vehicle design. 4) In propulsion, CFD has proven to be a valuable design analysis tool in a wide variety of situations. 5) Laminar-turbulent transition and fully developed turbulence are the pacing items of technology.

These achievements and prospects of CFD should be cause for encouragement, especially in view of the sporadic progress which is endemic to any field of science. Here is a classical example from aeronautics. The first attempt to calculate lift of a plate at incidence in an air stream was based on Newton's theory in 1726, and it was grossly underpredicted. More than a century later, Rayleigh proposed a theory which provided a slightly better approximation for lift, but was still not good enough. Three decades later, Rayleigh Joukowski, in 1907, theorized correctly the flow pattern about the plate. The lift formula derived from this theory predicted lift in close agreement with experiment. This, perhaps one of the most important evolutions of a theoretical concept in aeronautics, took about a century and a half.

CFD is still very young, and it does not have the breakthroughs that can be cited which are commensurate in importance to the discovery of the fundamental laws of nature. It has certainly assumed a new dimension equal in importance to theory and experiment in fluids. One hears sometimes the view that CFD usurps the place of theory. CFD has cut out its own place, and helps theory much the same way as physical experiments by providing insights into the physics. Furthermore, CFD may be used to validate the theory; after all, they have the governing equations in common.

VI. Challenges

A. Algorithms

Euler solutions as an asysmptotic limit of Navier-Stokes solutions, and the singularities of such solutions will always be a challenge for computations. Mathematical analysis of the Navier-Stokes equations leading to existence and uniqueness proofs in the case of the nontrivial three-dimensional flow-fields will provide realistic test beds for computational results. Mathematical results on the influence of small scales on large scales and vice versa and mathematical modelling of such interactions are crucial for dealing with turbulent flows.

Analysis of the discrete Navier-Stokes equations involving error bounds, error sources, stability and convergence proofs for truly multidimensional problems, and rigorous results on resolution requirements, artificial boundary conditions, etc., will be synergistic for leaps-and-bounds progress in CFD.

Improvement in efficiency of algorithms through convergence acceleration techniques, adaptive mesh techniques and methods for handling extremely stiff systems of equations, will allow computational solutions of problems with order of magnitude greater complexity than is possible now.

B. Engineering Applications

So far, the capability exists for the simulation of flow past an aircraft in level flight. The immediate challenge will be a full aircraft simulation including take-offs and landings. The simulation of the entire flight envelope of a military aircraft will be a greater challenge. Maneuverability and control aspects will require definition of new problem areas such as fluid dynamic control, fluid-structure interaction, etc. Such advances require an ability to predict turbulent flow for a wide range of conditions much more accurately than is possible today.

As Jameson [16] has pointed out, the integration of the predictive capability into an automatic design method using optimization, will require advances in algorithmic efficiency and computational power. It can possibly be realized in the next decade. Interactive calculations to optimize design appear to be feasible right now only for airfoil evaluation.

As pointed out in the NRC report, the applications and importance of CFD to hypersonic vehicle design is much greater than for the conventional aircraft. Simulations of hypersonic flow pertinent to upper atmospheric flight must take into account vehicle scale, flight velocity and altitude. In other words, the simulation should duplicate the Reynolds number, Mach number and total enthalpy, which is beyond the capability of the ground-based facilities. Thus, the applications and importance of CFD to hypersonic vehicle design is much greater than for the conventional aircrafts. Simulations for hypersonic aircraft will require an integrated approach involving interaction between external aerodynamics and reactive flow-fields of propulsive systems. Again, the ability to deal accurately with transitional and turbulent flows is crucial to success. This presents a great opportunity and challenge to CFD, and I am sure CFD will meet this challenge and convert even the nonbelievers.

C. Scientific investigations

Reactive flows pose yet another challenge to CFD. The scales in micromixing processes are much smaller than the turbulence scales themselves.

Nonlinear wave mechanics dominate laminar or turbulent flows. In fact it is believed that the nonlinear processes leading to turbulent spots in a laminar boundary layer are similar to the processes of turbulent bursts in turbulent boundary layers.

Turbulence is the grand challenge. It is an all pervasive ubiquitous phenomena present in, to cite a few instances, weather patterns, ocean currents, outer layers of the sun, ionosphere, high-temperature plasma, astrophysical jets, chemical reactors, and combustion. If CFD can solve the turbulence problem, the impact will be the same as the discovery of a new law of nature [37].

VII. Acknowledgement

The author is pleased to acknowledge the help of S. Chakravarthy, G. Erlebacher, A. Harten, A. Jameson, S. Osher, J. Thomas, B. van Leer, M. Salas, S. Zalesak and T. Zang in preparing this paper.

References

[1] J. P. Boris and D. L. Book, J. Comput. Phys. 11 (1973), 38.

[2] C. Canuto, M. Y. Hussaini, A. Quarteroni, and T. A. Zang, *Spectral Methods in Fluid Dynamics*, Springer-Verlag (1987).

[3] S. R. Chakravarthy, NASA Contractor Report 4043 (1987).

[4] A. J. Chorin, J. Comput. Phys. 2 (1967), 12-26.

[5] A. J. Chorin, Math. Comput. 22 (1968), 745-762.

[6] A. J. Chorin, J. Comput. Phys. 22 (1976), 517-536.

[7] M. Fortin, R. Peyret and R. Temam, J. Mech., V. 10 (1971), 357-390.

[8] S. K. Godunov, Mat. Sb. 47 (1959), 271-306.

[9] A. Harten and G. Zwas, J. Comput. Phys. 9 (1972), 568.

[10] A. Harten, AEC R and D Report, COO-3077-50, New York University (1974).

[11] A. Harten, Math. Comput. 32 (1978), 363-389.

[12] A. Harten, J. Comput. Phys. 49 (1983), 357.

[13] A. Harten and S. Osher, SIAM J. Numer. Anal., V. 24 (1987), 279.

[14] A. Harten, B. Engquist, S. Osher, S. Chakravarthy, J. Comput. Phys., V. 71 (1987), 231.

[15] F. H. Harlow and J. E. Welsh, Phys. Fluids 8 (1965), 2182-89.

[16] A. Jameson, AIAA Paper 87-1184 (1987).

[17] P. D. Lax and B. Wendroff, Comm. Pure Appl. Math. 13 (1960), 217-237.

[18] P. D. Lax, J. Stat. Phys., V. 43 (1986), 749-756.

[19] R.W. MacCormack, AIAA paper AIAA 69-354 (1969).

[20] A. Majda and S. Osher, Comm. Pure Appl. Math. 32 (1979), 797-838.

[21] G. Moretti, Computers and Fluids, 7 (1979), 191-205.

[22] J. von Neumann and R. D. Richtmyer (1950), vide R. D. Richtmyer and K. W. Morton, *Difference Methods for Initial-Value Problems*, Interscience, New York, 1967.

[23] R.-H. Ni, AIAA J. 20 (1982), 1565-1571.

[24] NRC Report, *Current Capabilities and Future Directions in Computational Fluid Dynamics*, National Academy Press, Washington, D. C. (1986).

[25] S. A. Orzag, J. Fluid Mech., V. 49 (1971), 75-112.

[26] S. A. Orszag and L. C. Kells, J. Fluid Mech. 96 (1980), 159-205.

[27] K. G. Powell, Vortical Solutions of the Conical Euler Equations, Ph.D. thesis, MIT, 1987.

[28] R. D. Richtmyer, vide R. D. Richtmyer and K. W. Morton, *Difference Methods for Initial-Value Problems*, Interscience, New York, 1967.

[29] P. L. Roe, Lecture Notes in Physics 141 (1980), 354-359.

[30] M. D. Salas, private communication.

[31] J. L. Steger and R. F. Warming, J. Comput. Phys. 40 (1981), 263-293.

[32] W. G. Strang, SIAM J. Numer. Anal. 5 (1968), 506.