View metadata, citation and similar papers at core.ac.uk
brought to you by CORE
provided by NASA Technical Reports Server

NASA Technical Memorandum 88356
USAAVSCOM Technical Report 86-A-2

AGARD Fluid Dynamics Panel Symposium on Applications of Computational Fluid Dynamics in Aeronautics

W.J. McCroskey

December 1986

(NASA-TM-88356) AGARD FLUID DYNAMICS PANEL N87-15181
SYMPOSIUM ON APPLICATIONS OF COMPUTATIONAL
FLUID DYNAMICS IN AERONAUTICS: PROCEEDINGS
(NASA) 19 p CSCL 01A Unclas
G3/02 40346

National Aeronautics and
Space Administration

United States Army
Aviation Systems
Command

NASA Technical Memorandum 88356

USAAVSCOM Technical Report 86-A-2

AGARD Fluid Dynamics Panel Symposium in Applications of Computational Fluid Dynamics in Aeronautics

W. J. McCroskey, Aeroflightdynamics Directorate, U. S. Army Aviation Research and
Technology Activity, Ames Research Center,
Moffett Field, California

National Aeronautics and
Space Administration

Ames Research Center
Moffett Field, California 94035

United States Army
Aviation Systems
Command
St. Louis, Missouri 63120

TECHNICAL EVALUATION REPORT

AGARD Fluid Dynamics Panel Symposium on APPLICATIONS of COMPUTATIONAL FLUID DYNAMICS in AERONAUTICS

W.J. McCroskey

U.S. Army Aeroflightdynamics Directorate
NASA Ames Research Center
Moffett Field, California

ABSTRACT

The Fluid Dynamics Panel of AGARD arranged a Symposium on "Applications of Computational Fluid Dynamics in Aeronautics," on 7-10 April 1986 in Aix-en-Provence, France. The purpose of the Symposium was to provide an assessment of the status of CFD in aerodynamic design and analysis, with an emphasis on emerging applications of advanced computational techniques to complex configurations. Sessions were devoted specifically to grid generation, methods for inviscid flows, calculations of viscous-inviscid interactions, and methods for solving the Navier-Stokes equations. The 31 papers presented at the meeting are published in AGARD Conference Proceedings CP-412 and are listed in the Appendix of this report. A brief synopsis of each paper and some general conclusions and recommendations are given in this evaluation report.

1. INTRODUCTION

The 58th Meeting of the AGARD Fluid Dynamics Panel (FDP) was held from the 7th to the 11th of April, 1986, in Aix-en-Provence, France. It included a major and timely symposium, organized by the recently established FDP standing committee on Computational Fluid Dynamics, with the following theme:

> "Computational Fluid Dynamics is making an increasingly major impact in aeronautical applications and on the aerodynamic design process. The rapid progress in computer capability, the general availability of large scale computers, and the parallel achievements in numerical analysis, algorithm development and user experience assure that the role of CFD in aeronautics will continue to expand.
>
> "The goal of the Symposium is to provide a balanced, if not exhaustive, assessment of the status of CFD in aerodynamic design and analysis. The emphasis in the symposium is on emerging applications of advanced computational techniques to complex and realistic configurations."

The Symposium spanned 3-1/2 days and consisted of five sessions concentrating on four major topics:

Session I.	Grid Generation
Session II.	Inviscid Flow I
Session III.	Inviscid Flow II
Session IV.	Viscous-Inviscid Interactions
Session V.	Navier-Stokes

As suggested by the titles of the individual sessions, the speakers addressed numerous detailed aspects of grid generation and a wide range of numerical methods and solutions. The titles of the 31 contributions are listed in Appendix A. Most of the papers in Session II were concerned with potential flows, while Session III was devoted to methods for solving the Euler equations. Both integral and finite-difference boundary-layer formulations were presented in Session IV. Most of Session V was devoted to three-dimensional solutions of the Reynolds-averaged Navier-Stokes equations with turbulence modeling (abbreviated throughout the Symposium as simply Navier-Stokes). In addition, a discussion period at the end of the Symposium elicited a wide range of informal comments by many of the attendees. The transcript of this discussion and the regular papers are published in AGARD Conference Proceedings CP-412. The members of the Program Committee for the Symposium are given in Appendix B of this report.

In keeping with the stated theme of the Symposium and with the general trends in the aeronautical community today, most of the papers dealt with complex configurations, such as wing-body-store combinations, control surfaces within jet nozzles, complete fixed-wing aircraft, missiles, and helicopter rotors and fuselages. In general, basic fluid dynamic phenomena, such as turbulence modeling or direct simulation of turbulence, were not considered; however, fundamental aspects of vortical flows and trailing-edge separation were treated by several authors. Also, research on basic numerical analysis and new algorithm development was less emphasized than the applications of existing/methods and/or their extensions to three dimensions.

It may be mentioned that the large majority of the papers were concerned with finite-difference and finite-volume methods, apparently reflecting the contemporary tendencies in most aeronautical companies and research laboratories. Also, it is noteworthy that, while most of the papers emphasized three-dimensional (3-D) flows, only three papers considered unsteady effects, and in one of these the unsteadiness in the solution arose as an unexpected complication in a problem with steady boundary conditions. However, the program committee had not solicited papers on unsteady flows, as these had been addressed in two previous AGARD meetings [Refs. 1,2] and in the associated reviews [Refs. 3,4]. Finally, hypersonic aerodynamics was discussed by only two speakers; this subject is the basis for an AGARD Symposium in April, 1987.

2. SYNOPSIS OF THE PAPERS

2.1. Grid Generation

The papers in Session I emphasized the great challenges that grid generation poses in successfully applying finite-difference and finite-volume methods to meaningful, practical aeronautical problems. Even defining the surface geometry accurately for complex 3-D bodies becomes a time- and CPU-consuming chore. Beyond that, creating meshes which conform to 3-D body surfaces and which distribute node points smoothly and efficiently throughout flow fields is also extremely challenging and laborious, especially if shock waves, contact discontinuities, and/or concentrated vortices are present. The use of interactive computer graphics is indispensible, and automation techniques are helpful, but considerable human intervention and ingenuity is required in all but the simplest cases. To paraphrase the authors in this session, grid generation for complex configurations can become more difficult and more expensive than obtaining the numerical solution itself. And, while a good grid will not guarantee a good solution, a bad grid will almost certainly produce a bad solution.

PAPER 1. OSKAM and HUIZING described a 2-D zonal grid generation method that is particularly suitable for multielement airfoils. Each element of the airfoil is embedded in one or more separate zones. Within each zone, the conformal transformation between physical and computational space is computed by a variational formulation that minimizes the deviation from the desired grid properties, e.g. cell areas and orthogonality, while allowing points to move along some of the zonal boundaries. This freedom of movement facilitates matching the grid interfaces between zones, and local surgery can be done to relocate topological singularity points that may occur at zonal corners. An example was given of a grid that would be suitable for Navier-Stokes computations for an airfoil-flap-slat combination.

PAPER 2. WEATHERILL, SHAW, FORSEY, and ROSE presented a zonal technique for 3-D bodies of complicated geometry such as complete aircraft, in which the flow domain in physical space is subdivided into many nonoverlapping "blocks." A feature of this method is that, in principle, the grid structure and/or topology in each block may be different and relatively arbitrary, according to what is most appropriate for that zone. However, grid lines are forced to match and pass smoothly across the interfaces. The grids are generated simultaneously in each block by solving sets of inhomogeneous elliptic partial differential equations. The authors candidly discussed some of the problems that they encountered, such as grid lines crossing over neighboring lines, the difficulties in clustering grid points arbitrarily, and slow convergence, and they described how these problems were ameliorated. Results were shown for a variety of aircraft and missile configurations which used from 9 to 240 blocks. The computer memory required to generate a complete grid system is about half that needed by the authors' flow solver for the Euler equations, whereas the CPU time is generally an order of magnitude less to generate the grid than it is to compute the solution.

PAPER 3. FRITZ also used the above multiblock concept and elliptic partial differential equation (PDE) solvers in his 3-D grid generation method for complete aircraft, but there are several differences. His grid lines are continuous across zonal boundaries, but the slopes are not matched. Grid points are generated in each block in steps: first along the perimeters, then on each surface, and finally within each block separately. However, the surface grids of all of the blocks are patched together before the complete grids are generated inside the individual blocks. The data management strategy allows large grids of several million grid points to be generated on computers with modest main memories. However, no figures for CPU times or input/output times were given.

PAPER 4. EDWARDS described both a procedure for defining the geometry of a fighter aircraft and a new, efficient grid-generation method. The former uses an existing computer-aided design/computer-aided manufacturing system (CAD/CAM), and an interactive graphics workstation to manipulate the input data base that defines some 50 components of an F-16 aircraft. It also constructs the appropriate surfaces for CFD modeling of the wing, fuselage, canopy, and faired-over inlet. The grid systems in the flow field are generated by applying "parabolic" difference operators in two of the three directions, to the elliptic PDEs (without the source terms) used by the previous authors. This approximation allows a noniterative, marching solution procedure to be used and thereby saves considerable CPU time and in-core memory, while retaining good control over the cell areas, clustering, and skewness. In the example given, the parabolic scheme was used to construct a relatively coarse grid over the whole flow field, and finer inner grids were interpolated to the desired distribution on the body.

2.2. Inviscid Flow I

This session was mostly devoted to linear and nonlinear methods for potential flows. These methods are particularly useful in the aeronautical design process, either because of their computational efficiency, or for their ability to treat complex configurations easily, or both.

PAPER 5. SMITH and WOODWARD assessed the capabilities and deficiencies of three established panel methods that are often used for linear calculations of complex geometries in supersonic flow. They compared panel solutions for relatively simple wings, bodies, and wing-body combinations with the results of experiments, higher-order panel methods, and Euler calculations. From this comparison, they identified the following areas where problems are most likely to arise:

1. Spurious reflections and other discrepancies at wing tips,
2. Rounded leading edges and other regions of high surface slopes,
3. The neighborhood of wing-body junctions.

Rounded leading edges that produce detached bow shock waves and source discontinuities at fuselage panel edges were indicated to be the most troublesome.

PAPER 6. LE, MORCHOISNE, and RYAN described a panel code under development that incorporates some new numerical techniques to speed up the calculations. They have developed a fast iterative technique using the method of steepest gradients, and a special way of introducing extra collocation points in regions of high curvature. This method was shown to be significantly faster than a direct integration technique, with the gain increasing as the number of panels increased. Some preliminary results were shown for a sphere, wing, and helicopter and transport fuselages.

PAPER 7. STRAWN and TUNG discussed the adaptation of an unsteady, 3-D full-potential code to compute the transonic flow near the tips of helicopter rotor blades. For this class of problems, the vortical wake is very important, but calculating its structure and position explicitly or directly is beyond the current state of the art in CFD. Therefore, two different methods were developed to couple approximate rotor wake models with the finite-difference code. The first formulation iterates between (1) the local blade lift which is calculated by the full-potential code for a given angle of attack distribution, and (2) the blade-surface inflow, or effective angle of attack, which is obtained from an approximate, integral wake method. The integral code also predicts the instantaneous blade motion. The second wake-modeling scheme allows concentrated vortices of predetermined strength to be introduced into the interior of the computational domain by means of a "split-potential" formulation of the finite-difference equations. Although this work is not complete, encouraging agreement with experiments was obtained for both methods.

PAPER 8. KAFYEKE presented a 3-D transonic small-disturbance code that uses grid embedding to predict the aerodynamic interference of wing-body-pylon-store configurations. The mean-surface approximation inherent in the small-disturbance formulation greatly simplifies the grid-generation problem by allowing cartesian meshes to be used everywhere, without conforming to the actual surfaces. Furthermore, fine grids around each component are embedded within an overall coarse grid, and the solution procedure cycles between the coarse and fine grid regions to accelerate the convergence. Citing the good agreement obtained with experiments and the modest computational requirements of the code when using approximately 200,000 grid points, the author downplayed the need for supercomputers in the aeronautical design process.

PAPER 9. VAN DER VOOREN, VAN DER WEES, and MEELKER described the computational aerodynamics integrated system called MATRICS (Multi-component Aircraft Transonic Inviscid Computational System) that is under development at the National Aerospace Laboratory. This system currently solves the full-potential equation using a multigrid method enhanced at National Aerospace Laboratory NLR, with provisions for approximately modeling vortex sheets, propeller slipstreams, and jet exhaust plumes. Extensions are underway to the Euler equations in subdomains, and eventually to the Navier-Stokes equation. MATRICS is designed with both scalar and vector computers in mind, but all the data are stored in the main memory without I/O (input/output) transfer, in anticipation of the large memories that future supercomputers will feature. In addition to the systems aspects, the authors reported on effects of boundary conditions, free-stream "consistency," artificial viscosity, and the size of the computational domain on the computed results for a representative transonic wing with and without a cylindrical body and/or propeller. They noted that the calculation of drag entails more stringent requirements for both grids and convergence tolerences than does lift.

PAPER 10. PETRIE and SINCLAIR gave a progress report on their development of a nonlinear field panel, or integral, method for solving the full-potential and Euler equations for complex configurations. In this approach, field computational grids are only needed in relatively small regions, and they need not be body-conforming. For compressible flows, however, nonlinear volume integrals arise that must be evaluated iteratively. No details of the solution technique nor of the computational requirements and efficiency were given.

PAPER 11. MARCHBANK presented an overview of the use of CFD in military aircraft design, including several examples of fighter aircraft in supersonic flow. He stressed three important requirements for effective use of CFD:

1. Speed - fast interaction with the aircraft geometry definition,
2. Utility - easy-to-use CFD procedures, oriented to the engineer,
3. Credibility - adequate accuracy of the aerodynamic results.

Panel methods more nearly satisfy the first requirement, both with regard to solution time and to geometry interface, whereas Euler calculations were found to be significantly more accurate for the supersonic flows of interest. Examples were given of drag optimization, wing design with drag constraints, pitching-moment optimization, and forebody and canopy design. These results were shown to lead to significant improvements in supersonic aircraft performance without compromising transonic characteristics, within short project timescales.

PAPER 12. WARDLOW and DAVIS began the sequence of papers on Euler methods that continued through Session III. Their method is a space-marching, finite-volume implementation of a Godunov-type scheme in supersonic flow. It should be explained that more conventional schemes assume a smooth (or mostly smooth), flow field, whereas the Godunov method considers the flow to consist of a series of piece-wise constant states with discontinuities occurring midway between mesh points [Ref. 5]. Analytic properties of 2-D shock waves and expansion fans are used to advance the solution across cell faces. In this paper, the addition of a central-difference predictor step and slope limiters gives second-order accuracy. This approach is more robust without explicitly adding artificial viscosity than are many conventional finite-difference schemes, although more computational work is required per grid point. Reasonable results were reported for a variety of finned tactical missile configurations.

2.3. Inviscid Flow II

Euler codes are emerging rapidly from the research stage, and as demonstrated in this session, they are finding increasing applications to 3-D configurations with strong nonlinear features. A variety of methods with a wide range of computer requirements were presented for solving the Euler equations.

PAPER 13. BREDIF, CHATTOT, KOECK, and WERLE used an explicit, multigrid finite-volume method to calculate the transonic and supersonic flow in an axisymmetric nozzle with control surfaces protruding into the stream just upstream of the nozzle exit. A relatively coarse grid system of approximately 60,000 nodes in two blocks was used, requiring on the order of 10 min. CPU time on a Cray 1S computer. The results were compared with linear theory and experiments; it was found that the thrust deflection caused by the control surface was accurately predicted, but the loss in thrust which was due to the interaction was underestimated.

PAPER 14. LEICHER employed a different explicit, multiblock, multigrid finite-volume method to calculate transonic flow over a wing and subsonic flow in the diffusers of a turbine and a wind tunnel. A wing-propeller combination was also simulated using an actuator-disk model at the plane of the propeller. Extensive grid-refinement and mesh-spacing studies were carried out for the wing, with the number of grid points varying from approximately 20,000 to 1,200,000. From the information given in the paper, the latter appeared to require over 100 hours of CPU time on an IBM 3083 computer, but an intermediate grid of about 150,000 points seemed to give satisfactory results in much less time. The results for the Kaplan turbine diffuser showed evidence of bimodal steady-state solutions.

PAPER 15. MURMAN and RIZZI provided one of the more fundamental and provocative presentations of the symposium. Their paper concerned the capabilities of Euler codes to generate and simulate leading-edge vortices on delta-wing configurations at high Reynolds numbers. Several examples were given of delta wings with sharp leading edges, in both subsonic and supersonic flow. Concentrated vortical structures emerged in the solutions with approximately the correct total pressure loss in the centers of the structures. From their own and other studies, they found this total pressure loss to be approximately independent of numerical parameters, such as grid spacing and the level of the artificial dissipation. Also, the calculated total pressure loss was relatively independent of whether or not physical viscous-stress terms were included in the equations. On the other hand, this loss was found to be sensitive to flow and geometry parameters, as it should be. Professor Murman explained his argument that "...any mechanism, whether real or artificial (i.e., numerical - author), which gives the vortex sheet some thickness will lead to a total pressure loss." He went on to discuss the structure of vortex cores, some implications of total pressure losses with respect to vortex breakdown, and solutions that were globally stationary but locally unsteady. Overall, this paper identified phenomena which will manifest themselves in future calculations and which will surely be the subject of further controversy and investigation.

PAPER 16. PERRIER, whose group at Avions Marcel Dassault-Breguet has pioneered the use of finite-element methods for aeronautical applications, gave a brief review and comparison of finite-volume and finite-element methods for solving the Euler equations. He expressed the view that Navier-Stokes solvers are too expensive for routine industrial use, but he stressed that potential methods fail to predict many nonlinear aerodynamic phenomena accurately enough. He showed numerous examples of solutions for complex flows and/or configurations, and he discussed the role of Euler codes as approximations to Navier-Stokes codes and as possible aids in understanding turbulence.

PAPER 17. EBERLE and MISEGADES described a new finite-volume Euler code that is claimed to be third-order accurate and which can be run in either an explicit or implicit mode. Their method also uses Godunov concepts (see Paper 12, above). A relatively simple grid-generation scheme with grid embedding allowed them to generate several impressive solutions, using up to 520,000 cells, for a complete fighter aircraft in transonic and supersonic flow (see Fig. 1. This figure was in color in the original paper). A limited grid-refinement study indicated that grids of this size were required for reliable results. The CPU times for the computations, which were done on Cray X-MP machines, were not quoted, but based on the number of grid points and the properties of the code, one might infer that they were several hours.

PAPER 18. KARMAN, STEINBRENNER, and KISIELEWSKI also presented a demonstration Euler calculation for a complete fighter aircraft, although their solution was not not fully converged at the time of the Symposium. They started with a multiblock, explicit finite-volume predictor-corrector formulation, and employed 20 blocks with a total of about 530,000 grid points. After 1500 iterations, they switched to an implicit flow-solver with a faster rate of convergence, but this scheme was not described. The results presented after 3100 iterations appeared to be evolving toward agreement with experimental data. In a private communication, the authors have since confirmed this trend after another 1000 iterations, although the experimental multiple-shock structure on the wing never materialized in the solution; this behavior is

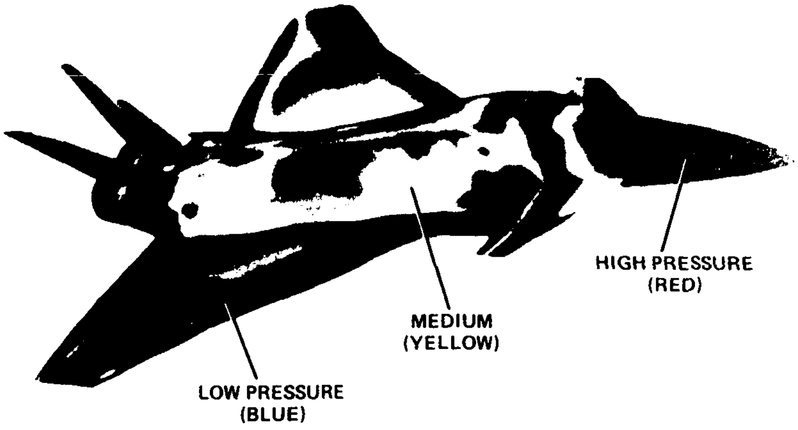

Fig. 1. Computed surface pressure distributions on a fighter aircraft; Paper 17.

attributed to inadequate grid resolution. The code was run on Cray X-MP and Cray 2 machines for a total of some 30-35 hours, of which approximately 30% was I/O time to the SDD (solid-state device) external memory device.

2.4 Viscous-Inviscid Interactions

As suggested by the title, the papers in Session IV addressed the relatively inexpensive introduction of viscous effects by means of boundary-layer concepts coupled with inviscid flow solvers. Concerning this approach, a brief explanation of the nomenclature used in this report may be in order. The coupling is called "weak" if the inviscid pressure distribution is impressed directly on the boundary layer and the resultant boundary layer weakly perturbs the inviscid solution. At least in 2-D steady flows, this procedure becomes singular in the neighborhood of flow reversal. "Strong" coupling means that the boundary-layer calculation is performed in an inverse mode and the computed pressure distribution is intimately and strongly coupled with the inviscid flow-solver. Many of the special techniques that have been introduced to accelerate convergence or to improve stability tend to blur this formal distinction between weak and strong coupling, but the "strong" terminology is retained herein when their intent is to achieve strong-coupling results.

PAPER 19. REIS and THOMPSON compared weakly-coupled calculations by two different numerical methods with measurements of 2-D trailing-edge separation. Their study included a laudable, careful assessment of numerical errors and uncertainties. However, they found that while the numerical errors were different, the discrepancies with the experiment were about the same; therefore, the turbulence model was blamed. Not surprisingly, computed surface pressures (lift) were more satisfactory than skin friction and wake profiles (drag).

PAPER 20. SCHMATZ and HIRSCHEL presented two zonal schemes for coupling the Euler equations with both the boundary-layer and Navier-Stokes equations, depending on the region of the flow field and the strength of the interaction. In the "alternating (weak) coupling" procedure of cyclic iteration through three distinct zones, different numerical schemes were used for each equation set, with the grids overlapping between the Euler and Navier-Stokes zones. Good results were shown for airfoils without separation. In the "close coupling" method, weak interaction of the boundary layer was retained on the front of the airfoil, but the Navier-Stokes grid near and behind the body was embedded in an outer Euler grid. The Euler and Navier-Stokes zones were solved simultaneously with a common implicit relaxation algorithm. In 2-D, a speedup of about a factor two was realized compared to a full Navier-Stokes solution; the gain in 3-D problems remains to be determined.

PAPER 21 was withdrawn, regrettably.

PAPER 22. GULCAT studied a 3-D unsteady boundary layer on a body placed impulsively into motion. He solved the incompressible laminar boundary-layer equations in physical space, eschewing similarity transformations. He employed a streamwise- and time-marching scheme that is implicit in the normal direction; the method seems to follow the spirit of Blottner's method [Ref. 6] for 2-D steady boundary layers. The resultant code is relatively simple, computationally efficient, and stable for nonsingular boundary-layer flow. He obtained good results for the flow-reversal time, pressure distribution, and pressure distribution for the impulsive flow past a circular cylinder. Solutions on the plane of symmetry of an oblate spheroid were also given up to the time of the development of flow reversal at the rear stagnation point.

PAPER 23. CLER presented a number of 3-D solutions obtained on helicopter fuselages, air intakes, and hub fairings. This effort was largely motivated by the need to estimate the drag of bluff bodies with large separated base flows. Most of the results were obtained with a panel code combined, but not directly coupled with, a 3-D integral boundary-layer code that located the lines of separation. While obviously not reproducing all the flow features, these solutions often help identify troublesome regions and suggest modifications to the body geometry. The crucial need for an adequate vortex-wake model was noted, and preliminary results were shown that had been obtained with the ONERA discrete-vortex method for a rotating blade in forward flight. However, this technique was found to be difficult to combine with the fuselage panel and boundary-layer programs, and much more development will be required to meet this objective.

PAPER 24. VAN DALSEM and STEGER described a simple and efficient algorithm for solving the unsteady, 3-D boundary-layer equations in either a time-accurate or relaxation mode. This approach is claimed to be more flexible and easy to apply than space-marching procedures. Their code switches from a direct (weak) to an inverse (strong) mode near and within reverse-flow regions, which permits separation to be computed readily. Two examples of good results were shown for prescribed pressure distributions. They also examined the use of this boundary-layer algorithm to speed up the convergence of an existing Navier-Stokes flow solver. The combination results in a so-called "Fortified Navier-Stokes" scheme that looks very promising. Fine-grid boundary-layer results near the wall are used as forcing functions in the thin-layer Navier-Stokes equations, which are solved only on a coarse grid. Two examples were given in which the accuracy of fine-grid Navier-Stokes solutions was obtained 20 times faster with the Fortified Navier-Stokes scheme.

PAPER 25. LAZAREFF and LE BALLEUR described their Multi-Zonal-Marching (MZM) method for turbulent, 3-D boundary layers with viscous-inviscid interaction. The MZM method is a procedure for extending Le Balleur's well-established and successful 2-D integral "semi-inverse" (essentially strong coupling) techniques to three dimensions. Careful attention is given to the characteristic cones of influence in each zone, and multiple sweeps can correctly couple different flow domains with a wide range of flow directions. The computations of swept wings, for example, start at a leading-edge stagnation point, from which multiple sweeps are made along the leading edge, but within a narrow streamwise zone, before the final streamwise and spanwise sweeps progress over the upper and lower surfaces. Limited results were presented of a wing calculation with strong viscous-inviscid coupling, but with no flow separation. However, most of the results were obtained in the direct boundary-layer mode for ellipsoids and a reentry-type lifting body at angle of attack. The MZM strategy allowed the calculations to be continued, with no viscous-inviscid coupling, over the whole body, even though crossflow-separation behavior developed on these bodies.

PAPER 26. FERMIN concluded this session with an outline of two RAE viscous-interaction wing codes and a description of several swept-wing applications. Transonic small-disturbance and full potential codes have been coupled with a 3-D integral boundary-layer method in the direct mode. The former code was used to design the new RAE M2155 low-aspect-ratio research wing that comprises a major code-validation exercise in progress, and wind tunnel results from the initial tests of this wing were presented. Fairly detailed comparisons were made with this experiment and with one of a transport-wing and body combination. Computations done with the Viscous Full Potential code, with direct (weak) coupling, were in excellent agreement with the measurements in the subsonic case. However, transonic conditions with strong viscous-inviscid interaction were a major challenge. It was difficult to obtain converged solutions, and the computed pressure distributions were deficient in local regions, particularly when the boundary layer was close to separating. Suggestions for improvements included a different coupling scheme, the use of an inverse (strong) method, and using Euler codes for the inviscid flow.

2.5 Navier Stokes

The increasing availability of supercomputers and the progress in algorithm and software development have brought 3-D Navier-Stokes calculations of aeronautical configurations close to the realm of near-term reality. The papers in this session gave valuable insights into the general state of the art today, and into the developing trends in applied computational aerodynamics for viscous problems.

PAPER 27. SHANG outlined the most common approaches used in the aerospace community and reviewed a number of recent accomplishments, including simulations of flows around aircraft wings, fuselages, afterbodies, and inlets. He briefly discussed numerical efficiency and accuracy, boundary conditions, turbulence models, grid generation, data structure and management, and post-processing and display of results. He noted that the grid-generation phase occupies an ever-increasing fraction of the total elapsed time of numerical simulations, and he lamented the vast amount of information that is discarded during or soon after a typical investigation. The growing and lasting importance of "interdisciplinary computational fluid dynamics," in which Navier-Stokes equations are coupled with the governing equations of solid mechanics, chemistry, combustion, electromagnetics, optics, etc., was also stressed.

PAPER 28. WAI, BLOM, and YOSHIHARA employed the Parabolized Navier-Stokes (PNS) method, in which the streamwise-diffusion terms in the equations are neglected, to calculate the supersonic flow over a generic fighter aircraft configuration at high-lift conditions. They used an existing PNS code, described in the next paper, with local regions of subsonic flow computed by an unsteady Navier-Stokes code. Data for comparison were very sparse, but reasonable agreement of the numerical and experimental results was obtained. The calculation time for approximately 4100 grid points in the crossflow planes and about 1000 streamwise stations was 5.4 hours on an ETA 205 computer for a wing, fuselage, canard, and nacelle combination. A major fraction of this time was consumed in regenerating the grid at each streamwise station.

PAPER 29. CHAUSSEE discussed the NASA-Ames PNS code and several applications to supersonic and hypersonic configurations with important viscous effects. The parabolized approximation assumes that the inviscid flow is supersonic in the streamwise direction, and that the subsonic flow in the viscous sublayer is always positive in the streamwise direction. This allows the solution to march streamwise, giving substantial savings in CPU time, while an implicit algorithm is used to solve the flow in each crossflow plane. Crossflow separation is permitted, but streamwise separation is not. An algebraic eddy-viscosity model is used in turbulent cases. Examples included ogive-cylinders, sphere-cones with flaps, finned projectiles, the Space Shuttle orbiter, and the supersonic fighter which is the subject of Paper 28.

PAPER 30, Part I. FLORES, HOLST, KAYNAK, GUNDY, and THOMAS described the development of a zonal Euler/Navier-Stokes code called Transonic Navier-Stokes (TNS). The method uses a diagonalized implicit algorithm to solve the unsteady equations, but with a variable time step for faster convergence. They divided the flow into four zones for wings: two inner blocks near the surface where the thin-layer Navier-Stokes equations are solved, and two outer Euler blocks. As in the preceding two papers, turbulence is simulated by means of an algebraic eddy-viscosity model. Results for an airfoil in a solid-wall wind tunnel and for two transonic wings were given; the agreement with experiments was generally good, although some differences in regions of shock-induced separation were noted. CPU times on a Cray X-MP computer were on the order of one hour for 150,000 grid points.

PAPER 30, Part II. CHADERJIAN applied the aforementioned TNS code to a swept NACA 0012 wing at high angle of attack, with a view toward demonstrating the robustness, efficiency, and accuracy of the code, and assessing the characteristics of the grid. Major improvements near grid singularities were accomplished by computing the transformation metrics consistently to preserve a uniform free stream. Results were obtained up through maximum lift at M = 0.5 and 0.8, although data were not available for comparison. The author suggested improvements in terms of additional grid refinement, an improved numerical-dissipation model, and a turbulence model more suited to shock-induced separation.

PAPER 31. KORDULLA, VOLLMERS, and DALLMANN reported on their simulation of transonic laminar flow past a hemisphere-cylinder at angle of attack, and they analyzed the topology of the separated flow in great detail. Their time-accurate, finite-volume method uses a variation of the MacCormack explicit-implicit predictor-corrector scheme, and up to about 230,000 grid points were used in the present study; CPU times on the Cray 1S computer used were not quoted. The fine-mesh solution of a nominally steady case exhibited unsteadiness in the separated region, and the question of whether this was due to fluid physics or to numerical errors was not resolved. The main thrust of the paper, however, was the analysis of the topology of the computed flow field. Guided by earlier work of the third author and others on topological kinematics and bifurcation theory, considerable detail was extracted from the solution. Impressive sketches of complex vortical structures and of the major singular stream surfaces were presented (see Figs. 2 and 3).

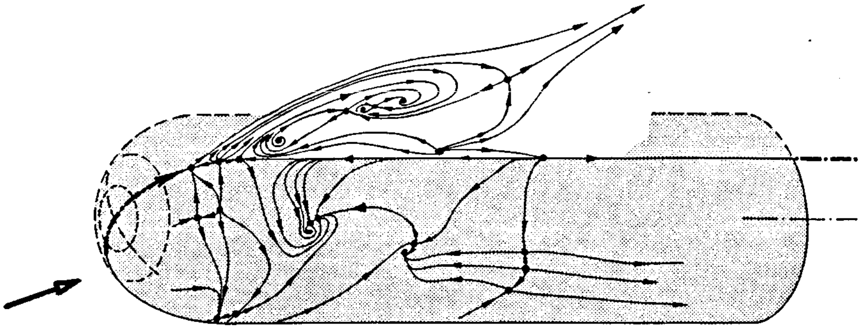

Fig. 2. Sketch of the major singular points in the symmetry plane and on the wall of a hemisphere cylinder at angle of attack in transonic flow; Paper 31.

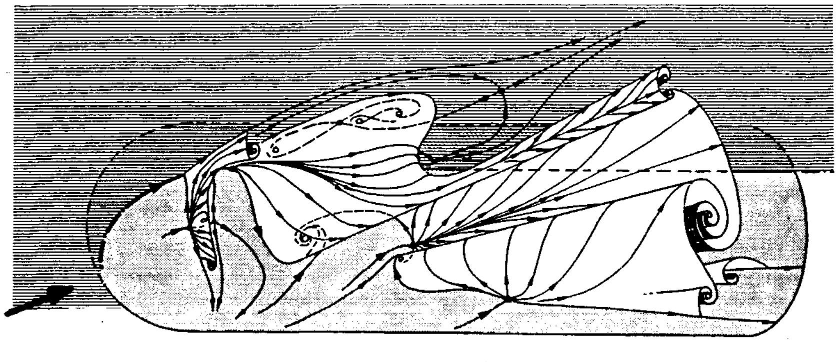

Fig. 3. Sketch of the major singular stream surface in the flow field of a hemisphere cylinder; Paper 31.

PAPER 32. Finally, SHANG discussed his pioneering computations of the flow field of the complete XC-240 experimental aircraft at a Mach number of 5.95. The equations in weak conservation-law form were solved in a single computational block using MacCormack's explicit unsplit algorithm, with the aid of a sophisticated data management scheme and the Solid State Device (SSD) of the Cray X-MP. The solution time for 475,200 grid points was about 20 hours, of which about 25% was consumed by I/O transfer to the SSD. Good agreement was obtained between computed and measured lift and drag, and on the whole, the pressure distributions and heat transfer were satisfactory. A major effort to display the results with the aid of 3-D color graphics produced some spectacular pictures of the surface streamlines, pressures, and temperatures.

3. DISCUSSION AND EVALUATION

The goal of the Symposium, as noted in the Introduction, was to provide an "... assessment of the status of CFD in aerodynamic design and analysis," with an emphasis on "... emerging applications of advanced computational techniques to complex and realistic configurations." Indeed, the material presented at the Symposium was generally indicative of the state of the art of applied computational aerodynamics in the NATO countries, in terms of current activities, methods, numerical and physical modeling employed, and computers in service.

However, the Symposium had definite limits in scope and content. The Program Committee was obliged to leave out noteworthy efforts in almost every country, and there was little representation from Sweden and none from Japan. Also, research on new algorithm development and turbulence modeling was generally excluded, apparently by intent. Beyond that, it was somewhat surprising that the work of Anthony Jameson and his colleagues on the FLO-xx series of codes and their many permutations was hardly represented.

As a general comment on Session IV, Viscous-Inviscid Interactions, weakly-coupled 3-D methods have been developed in several countries over the past few years, but these efforts were not well represented. It is not clear whether this was accidental, or whether it reflects a new trend toward developing strong-coupling methods that simply has not yet materialized.

The Symposium did make clear the widespread use of CFD throughout the aeronautical industry, and it revealed that the spectrum of methods in use is growing rapidly to include Euler and Navier-Stokes codes applied to complex configurations. The Symposium was also very timely, because the emphasis has clearly shifted from 2-D problems to the more realistic world of 3-D. And as discussed in Section 3.1.2 below, the advent of this geometrical complexity is causing the time, manpower, and computer costs of grid generation and data analysis to become much more significant relative to the effort previously involved in simply generating solutions.

3.1 Costs and Capabilities

Today the aerodynamicist has a wide range of computational methods available, at a correspondingly wide range in manpower and computer costs. As mentioned in the Introduction, finite-difference or finite-volume methods remain much more popular for nonlinear problems than for field-panel or finite-element methods. M. Pierre Perrier argued persuasively for the latter, but no direct quantitative comparisons were made, nor could be made from the material presented at the Symposium. For inviscid flows, 3-D potential codes continue to be developed and improved, but this approach is now relatively mature and the development of Euler methods has become more fashionable. However, the treatment of viscous effects has not kept pace, at least in 3-D. Viscous/inviscid coupling methods are potentially very cost-effective in CPU time, but much remains to be done to make them work properly for flows with strong shock waves. Pioneering showcase Navier-Stokes solutions are emerging rapidly; but as noted below, the CPU times remain very long, the convergence is slow, and questions about accuracy, the effects of grid refinement, and turbulence modeling have barely been asked, let alone answered.

3.1.1 Computational Requirements. The different levels of approximations to the governing equations of fluid mechanics, and the associated difficulties in solving them numerically, are well known and need not be reiterated here. However, it is instructive to examine briefly the dominant factors that determine the computational requirements for the range of methods that were presented at the Symposium. Of course, many factors such as the degree of maturity of different algorithms and the impact of new computer architectures are difficult to quantify; but the approximate CPU time required for most of the methods can be crudely estimated with the aid of the following formula:

$$\mathrm{CPU} = A \times W_{GI} \times N_G^m \times N_I/\mathrm{FLOPS} \tag{1}$$

where: A = "numerical inefficiency" factor

W_{GI} = number of operations per grid point per time step

N_G = number of grid points or panels

m = 1 for finite-difference , $\approx$ 2 for panel methods

N_I = number of iterations, or number of time steps for an unsteady calculation

FLOPS = number of floating-point arithmetic operations per second

The inefficiency factor, A, is introduced here to emphasize that few codes take full advantage of the computer being used; in practice this factor is a function of the programming efficiency, the degree of vectorization, the coupling between the grid and the solution algorithm, the data-management strategy, the user experience, etc. Ideally, its value should approach unity; but especially with the advent of supercomputers with novel architecture, or with I/O transfer to external memory devices, it could well be 2.0 or even larger.

The number of arithmetic operations per grid point per iteration, W_{GI}, is a strong function of the numerical method; that is, of the flow equations, the boundary conditions, the solution algorithm, and whether the grid metrics are computed at each time step or stored in memory. The quantity N_G represents the number of grid points for a finite-difference method, the number of elements for a finite-element method, or the number of panels for a panel method. Consequently, $W_{GI}\ N_G$ represents the number of arithmetic operations that must be performed at each iteration or time step, although in some instances with panel methods, $N_G \log N_G$ is a more accurate representation than N_G^m.

The total number of iterations, N_I, to converge to the desired level of accuracy (or the total number of time steps in an unsteady problem), is generally the most difficult quantity to estimate. And unfortunately, N_I for a given algorithm is often both problem-dependent and grid-dependent. This is particularly true in the early stages of maturity of a code, such as many of those presented in Sessions III - V. The value of N_I is generally significantly greater for explicit methods than for implicit ones; but on the other hand, W_{GI} for implicit methods tends to be greater.

Finally, the sustained computing speed, FLOPS, is a function of the computer clock speed and architecture, the data structure and techniques of the code, and the memory requirements (in-core or external memory). The aerodynamicist has relatively little control over this quantity; rather, it is largely dictated by management, i.e., by the size and cost of the computer system.

It is clear from Eqn. 1 that many different factors determine the CPU time, and hence, the computational cost, of an aerodynamic calculation. Unfortunately, none of the papers provided all of the ingredients specified in Eqn. 1; however, enough information was provided either at the Symposium or elsewhere, or could be inferred, to establish some trends and order-of-magnitude estimates. This information is summarized in Table 1, for a hypothetical wing-body combination of moderate geometrical complexity. Here the Mach number is implicitly assumed to be subsonic in the linear case, supersonic for the Parabolized Navier-Stokes estimate, and transonic otherwise; and the reference computer speed is that of the Cray 1-S, i.e. FLOPS = 80×10^6.

It should be emphasized that these estimates are very approximate, hypothetical, and somewhat abritrary; therefore, they could easily be off by a factor of 2 or more. Also, they reflect the arguable premise that increasing numbers of grid points should be accompany the increasing sophistication in the flow modeling, in order to capture the more complex flow physics that motivate the more complex approaches. Nevertheless, Table 1 gives a general, qualitative picture of the differences in the overall computational requirements of many of the approaches that were described at the Symposium. It indicates, for example, that transonic small-disturbance, full-potential, and PNS codes can be more or less competitive with panel codes, with respect to computer resources. The table also gives some idea of the price one would have to pay today to obtain full viscous simulations of transonic flow. These observations deserve further comment, and this leads logically to the question of what capabilities the different methods actually offer for aerodynamic design and analysis.

Table 1. APPROXIMATE COMPUTATIONAL REQUIREMENTS
FOR A COMPLEX WING-BODY COMBINATION

Method	W_{GI}	N_G, millions	N_I	CPU, minutes	Total memory, words $\times 10^6$
Linear (panel)		(1000 panels)		2 - 20	0.5 - 1.0
Transonic Small Disturbance	100	0.1-0.2	100 - 300	5 - 15	0.5 - 1.0
Full Potential	500	0.1-0.2	200 - 500	10 - 30	1 - 4
Euler	1000-3000	0.2-0.5	500 - 5000	50 - 500	2 - 10
Parabolized Navier-Stokes		$N_G \times N_I \sim 2\text{-}5 \times 10^6$		10 - 60	0.5 - 1.0
Navier-Stokes	1500-4500	0.5-2	1500 - 10000	1000 - 5000	15 - 60

3.1.2 Capabilities and Limitations. Linear panel codes have been the workhorses of the computational aerodynamics community for many years, and they are generally considered to be the most flexible and least inexpensive approach for complex configurations. They are inappropriate, however, for transonic flows; and as noted above, nonlinear potential codes have been developed to a competitive position costwise. Additional difficulties can arise in supersonic flows, cf. Paper No. 5; therefore, the estimates of computer resources in Table 1 suggest that space-marching Euler or PNS codes might well be attractive alternatives for supersonic cases. However, these nonlinear codes have had less time to benefit from user experience, and they require more effort to generate the grids. Consequently, they would probably be more difficult to use today in an engineering environment.

Today, almost any company, organization, or institution that has significant connections with transonic aircraft or missiles routinely uses nonlinear 3-D potential codes, and the helicopter community is moving rapidly in that direction. The recent trend to move on to Euler codes seem to be motivated by three main factors: first, to treat stronger shock waves than is permitted by the potential formulation; second, the ability to capture vortex sheets and other aspects of rotational flows; and third, to build a bridge to Navier-Stokes codes. The results of AGARD Working Group WG-07 [Ref. 7] are especially noteworthy with regard to the first consideration, and Papers 14, 15, and 16, as well as Ref. 7, provide representative demonstrations of the second factor.

Two points can be made regarding the nonlinear inviscid codes, especially the Euler ones. With the neglect of viscous effects, the calculated lift is too high, the drag is too low, and the shock wave is in the wrong position; consequently, the pitching moment is generally incorrect, too. In practice, these effects may be masked or counterbalanced by the effects of coarse grids, numerical dissipation, and other numerical errors (as illuminated by Paper 14, for example). The net effect is to introduce an element of uncertainty in the results, which can seldom be evaluated short of performing extensive, and expensive, grid-refinement studies. Similar criticisms can be leveled at the Navier-Stokes calculations as well, as discussed below.

Secondly, the rise in applications to complex 3-D configurations has dramatically increased the importance of both grid generation and post-processing data analysis, to such an extent that now they are often the pacing items with respect to manpower and elapsed time. As Dr. Wolfgang Schmidt noted in the closing discussion period, it may take months to set up the complex mesh for a complete aircraft, followed by a day or two of "clock" time to complete an hour or so of CPU time, and it may then take months to "reduce" the data fully. (The comparison with large wind-tunnel projects goes without saying.) Furthermore, this added complexity means that it is becoming increasingly difficult to hand-off these powerful nonlinear codes to inexperienced users.

Coupled viscous-inviscid interaction methods are generally considered to be promising for design applications because of their computational efficiency relative to Navier-Stokes approaches. Two-dimensional formulations linking inviscid and viscous algorithms formed the basis of a major AGARD Symposium in 1980 [Ref. 8]. These methods have matured and come into general use since then, including successful applications to flows with small amounts of separation. Several successful extensions to 3-D have appeared in the recent literature, although these are mostly weak-coupling methods applied to unseparated flows. However, the confidence level that exists for the 2-D methods seemed to be lacking at this Symposium. As noted above, this may reflect a period of renewed effort to develop new strong-coupling methods that are robust and efficient, and as noted by Fermin in Paper 26, which give accurate predictions of drag. The ensemble of papers in Session IV suggests that much work remains to be done, even for high-aspect-ratio wings, when strong coupling is required between the viscous and inviscid regions. The new RAE experiments discussed in Paper 26 should provide good targets and challenges for improvements in this area.

Within the scope of the Reynolds-averaged formulation of the Navier-Stokes equations, the limitations of the preceding methods are theoretically eliminated. The computational requirements shown in Table 1 indicate that this approach is not yet practical for routine aeronautical analysis and design, although this conclusion would completely change within a decade if the past trends in computer technology and algorithm development are extrapolated into the future [Ref. 9]. However, the primary issue here is the validity of the results that are being computed today. In most cases, 2-D Navier-Stokes calculations are approximately as good as the average wind-tunnel results [Ref. 10]. However, the issue of 3-D code validation is mired in controversy and uncertainty regarding turbulence modeling, spatial resolution, the effects of numerical dissipation and other numerical errors, and the completeness and reliability of the relevant experiments.

For most relatively benign flows, the Navier-Stokes calculations agree reasonably well with experiments (as do the results of the simpler codes at a fraction of the cost), provided "reasonable" grids are used and some attention is paid to the numerical dissipation parameters. However, in those cases where the Navier-Stokes approach is fully justified over simpler methods because of strong viscous-inviscid interaction and/or massive flow separation, virtually all of results to date have showed significant discrepancies when detailed comparisons were made with experiments. In the opinion of most turbulence experts, the simple turbulence models used in all the large Navier-Stokes codes described at the Symposium are inadequate for such applications, and in the opinion of the leading numerical analysists, the grids would have to be refined considerably before the solutions could be expected to be grid-independent.

Therefore, one may conclude that today's 3-D Navier-Stokes solutions are useful but largely qualitative. However, the extent of the information buried in the results is enormous, and they probably contain valuable physical insights that may not be evident in experimental data. The issues of turbulence modeling and grid resolution are, in fact, being addressed vigorously, and the papers in Session V give a preview of some things expect in the future.

3.2 Additional Problems and Issues

In addition to the issues of costs and capabilities of the various CFD approaches, the topics listed below emerged either during the formal sessions or in the final discussion period.

3.2.1. Adaptive Grids. None of the regular papers discussed solution-adaptive grids. However, this concept offers the potential of reducing the computational requirements by placing grid points where they are most needed and making more efficient use of the computer resources available. Several participants indicated that this topic is under active investigation.

3.2.2. Turbulence Model vs. Numerical Errors. When numerical solutions are suspected to be in error, one may question either the turbulence model or the numerical model. This issue has several components. The first is the relative magnitude of the numerical viscosity that is inherent in most CFD methods compared to the laminar or turbulent viscosity; one may ask which of these is the dominant factor in different parts of the flow field. The numerical viscosity normally varies in proportion to the grid spacing, and thus is a property of the numerics, whereas the physical viscosity is a property of the fluid and/or flow gradients. In the Euler calculations of Murman and Rizzi, for example, the numerical viscosity is the mechanism which permits the formation of shear layers in a numerical simulation of an inviscid flow. However, they claimed that any amount of numerical viscosity, however small, was sufficient to allow a realistic solution to develop, and that their results were insensitive to its value. In some of the Navier-Stokes calculations, (Papers 27 and 30, for example), the numerical viscosity was rather large in some parts of the flow field, especially near shock waves, but was probably small compared to the turbulent viscosity in the interior of the boundary layer. Unfortunately, it has not yet been possible to refine these grids enough to determine how sensitive these 3-D solutions are to numerical viscosity, or other numerical parameters.

Another issue, raised by Dr. Tuncer Cebeci, is the grid resolution in the direction normal to the wall that is required for accurate prediction of the surface shear stress. Boundary layer methods tend to use many more points across the viscous layer than Navier-Stokes methods, although the trends are to use comparable spacing immediately adjacent to the wall. Again, more grid-refinement studies are needed.

With the exception of Papers 19, 25, and 26, all of the turbulent calculations were done using eddy-viscosity models for simplicity. As noted in Section 3.1.2, this would be a source of serious error for many complex flows, in the opinion of most turbulence experts. However, this issue was not a major theme of the Symposium, nor an overriding concern of most of the participants.

3.2.3. Drag Calculations. This topic was given only secondary treatment in almost all of the formal papers. However, in the discussion period the accuracy of drag predictions emerged as being very important to many participants, and it seems certain to receive more attention in the future. Mr. J. W. Slooff stated, with reference to the large number of colored graphs of results that were presented, that an improvement in drag prediction of 10 counts would be worth more than a thousand pictures.

3.2.4. Highly-Vortical Flows. A major motivation for many of the advanced CFD methods under development today is the desire to compute the entire flow field of flight vehicles at high angles of attack, and these flows contain complex vortical structures. Such flow features present major challenges in grid generation and computation, as many existing techniques are not suitable for capturing and preserving the strong flow gradients that are involved. Solution-adaptive grid techniques may be advantageous here. Also, new techniques of capturing or fitting concentrated vortices may have to be developed, by analogy with the successful efforts in the past to develop both shock-fitting and shock-capturing methods.

This class of flows also provides major challenges in analyzing and understanding the computed results, as the topologies of vortical structures associated with 3-D separation are very complex. Paper 31 included some exciting examples of this complexity, which requires extensive computer-graphics capability to even begin to analyze the flow structure.

3.2.5. Future Topics. The fixed-wing aircraft industry has been the primary driving force for the computational aerodynamics techniques that were presented at this Symposium. As CFD methods for this class of vehicles mature, one may expect to see expanded applications to rotorcraft and V/STOL aircraft, turbomachinery, and hypersonic vehicles. Participants also suggested that major future efforts may include transition prediction; unsteady aerodynamic flows; advanced turbulence modeling, including numerical simulations of turbulence; and optimization of body shapes to satisfy prescribed aerodynamic properties, such as pressure distributions.

Finally, in Paper 27, Dr. J.S. Shang discussed at length "interdisciplinary computational fluid dynamics," in which fluid-flow equations (Navier-Stokes equations, in his perspective) are coupled with the governing equations of solid mechanics, chemistry, combustion, electromagnetics, optics, etc.

4. CONCLUSIONS AND RECOMMENDATIONS

Computational Fluid Dynamics has established a firm role in aerodynamic design, and this Symposium revealed the wide spectrum of CFD tools that are now available to the aeronautics community. Furthermore, the application of these tools to problems of ever-increasing complexity is accelerating rapidly, and computer hardware is advancing even faster than improvements in numerical algorithms. As a result, the users of modern CFD technology can generate vast amounts of information, and their managers are not always sure how valid or useful this mountain of data really is.

Three important needs arise from this situation. The first is to raise the confidence level of CFD results, as opposed to overselling their importance or covering up their limitations. That is, we urgently need better validation of the codes and convincing demonstrations of their capabilities and limitations. This requires more careful grid-refinement studies and numerical-error checks, leading to the establishment of reliable error bands on CFD results. Uncertainty analysis has become an important part of quality experimental research and testing, and there is no reason not to establish similar quality control for CFD. It is also essential to establish more standard test cases and to compare results for them. A limited number of new experimental configurations should be established with the specific needs of CFD code validation in mind, and with more redundancy, higher accuracy, and better flow quality than has been the norm heretofore. Numerous AGARD working groups and other parties have performed valuable services of this type in recent years, by defining test cases, coordinating the experimental efforts at several different laboratories, and assessing the results. This kind of effort should be continued with renewed vigor.

Second, better data and information management tools are becoming as important as better computing hardware for post-processing the CFD results and extracting the valuable information that they contain. Although not all of the Symposium participants shared the writer's enthusiasm for color computer graphics, they all agreed that this aspect of applied CFD is essential and is growing in importance. This is another area in which AGARD should try to facilitate better multi-national cooperation and information exchange.

Third, the issue of turbulence modeling continues to be a cloud over CFD results for viscous problems. Although this issue was not a major theme of this Symposium, it will probably hinder both progress in and acceptance of applied computational aerodynamics for many years. On the one hand, CFD researchers must forge ahead in the development of new methods, without waiting for the fundamentals of turbulence to be understood fully. However, they need to structure their codes as much as possible to accept new turbulence models, and they must be willing to incorporate them as better ones become available.

In conclusion, the Symposium provided a timely and valuable forum for exchanging information about recent developments in applied computational aerodynamics, and it clearly fulfilled the stated goal of assessing the status of CFD in contemporary aeronautical design and analysis. Together with the problems and issues were that were identified, many successes of CFD in recent years were highlighted, and the trends of current developments point to a promising future.

In the near term, refinements in the viscous-inviscid methods should overcome many of the shortcomings of present inviscid methods without significant additional costs, and continuing improvements in computer hardware and software and in numerical algorithms will create more demand for the Navier-Stokes codes. Thus the present level of maturity of inviscid computational aerodynamics can be expected to extend to viscous flows within a few years for fixed-wing aircraft. At the same time, CFD should find rapidly-increasing applications in other fields of aeronautics, such as rotorcraft and turbomachinery. In the longer term, it seems clear that CFD will be combined with a wide range of other disciplines to expand greatly its range of applications and usefulness.

5. ACKNOWLEDGEMENTS

Special thanks are due to Dr. Joseph Steger for his valuable comments and suggestions during the preparation of this Evaluation Report.

6. REFERENCES

1. Transonic Unsteady Aerodynamics and Its Aeroelastic Applications, AGARD CP 374, Toulouse, Sept. 1984.
2. Unsteady Aerodynamics - Fundamentals and Applications to Aircraft Dynamics, Gottingen, AGARD CP 386, May, 1985.
3. Mykytow, W. J., Review of SMP 1984 Symposium on "Transonic Unsteady Aerodynamics and Its Aeroelastic Applications," AGARD CP 371, Addendum 1, June 1985.
4. Mabey, D. G., and Chambers, J. R., Technical Evaluation Report on "Unsteady Aerodynamics - Fundamentals and Applications to Aircraft Dynamics," AGARD Advisory Report No. 222, Jan. 1986.
5. Richtmyer, R. D., and Morton, K. W., Differential Methods for Initial-Value Problems, 2nd edition, Interscience, New York, 1967, pp. 338-345.
6. Blottner, F. G., "Chemical Nonequilibrium Boundary Layer," AIAA Journal, Vol. 2, No. 2, pp. 232-240, Feb. 1964.
7. Test Cases for Inviscid Flow Field Methods, AGARD Advisory Report No. 211, Yoshihara, H., and Sacher, P., editors, May, 1985.

8. Computation of Viscous-Inviscid Interactions, AGARD CP 291, Colorado Springs, Oct. 1980.

9. Peterson, V. C., and Arnold, J. O., "The Impact of Supercomputers on Experimentation: a View from a National Laboratory," Proc. ASEE Annual Conference on Computer Aided Engineering, Vol. III, pp. 1388-1401, Atlanta, June 1985.

10. McCroskey, W. J., Baeder, J. D., and Bridgeman, J. O., "Calculation of Helicopter Airfoil Characteristics for High Tip-Speed Applications," J. American Helicopter Society, Vol. 31, No. 2, pp. 3-9, Apr. 1986.

APPENDIX A. List of Papers - AGARD Conference Proceedings No. 412.

Session I - GRID GENERATION

1. B. Oskam and G. H. Huizing, "Flexible Grid Generation for Complex Geometries in Two Space Dimensions Based on Variational Principles"
2. N. P. Weatherill, J. A. Shaw, C. R. Forsey, and K. E. Rose, "A Discussion on a Mesh Generation Technique Applicable to Complex Geometries"
3. W. Fritz, "Numerical Grid Generation around Complete Aircraft Configurations"
4. T. A. Edwards, "Geometry Definition and Grid Generation for a Complete Fighter Aircraft"

Session II - INVISCID FLOWS I

5. J. S. Smith and D. S. Woodward, "An Assessment of the Use of Low-Order Panel Methods for the Calculation of Supersonic Flow"
6. T. H. Le, Y. Morchoines, and J. Ryan, "Techniques Numeriques Nouvelles dans les Methodes de Singularites pour l'Application a des Configurations Tridimensionnelles Complexes"
7. R. C. Strawn and C. Tung, "The Prediction of Transonic Loading on Advancing Helicopter Rotors"
8. F. Kafyeke, "Prediction of Wing-Body-Store Aerodynamics Using a Small Perturbation Method and a Grid Embedding Technique"
9. J. van der Vooren, A. J. van der Wees, and J. H. Meelker, "MATRICS - Transonic Potential Flow Calculations about Transport Aircraft"
10. J. A. H. Petrie and P. M. Sinclair, "Applications and Developments of Computational Methods for the Aerodynamic Problems of Complex Configurations"
11. W. R. Marchbank, "The Integration of Computational Fluid Dynamics into the Military Aircraft Design Process"
12. A. B. Wardlow and S. F. Davis, "A Second Order Godunov Method for Tactical Missiles"

Session III - INVISCID FLOW II

13. M. Bredif, J. J. Chattot, C. Koeck, and P. Werle, "Simulation d'un Systeme de Deviation de Jet a l'Aide des Equations d'Euler"
14. S. Leicher, "Numerical Simulation of Internal and External Inviscid and Viscous 3-D Flow Fields"
15. E. M. Murman and A. Rizzi, "Applications of Euler Equations to Sharp Edge Delta Wings with Leading Edge Vortices"
16. P. Perrier, "Utilisation des Codes Euler pour Calculs en Aerodynamique non Lineaire"
17. A. Eberle and K. Misegades, "Euler Solution for a Complete Fighter - Aircraft Configuration at Sub- and Supersonic Speed"
18. S. L. Karman, Jr., J. P. Steinbrenner, and K. M. Kisielewski, "Analysis of the F-16 Flow Field by a Block Grid Euler Approach"

Session IV - VISCOUS-INVISCID INTERACTIONS

19. L. Reis and B. E. Thompson, "Comparison of Finite Difference Calculations of a Large Region of Recirculating Flow near an Airfoil Trailing Edge"
20. M. A. Schmatz and E. H. Hirschel, "Zonal Solutions for Airfoils Using Euler, Boundary-Layer and Navier-Stokes Equations"
21. withdrawn

22. U. Gulcat, "Numerical Investigation of the Laminar Boundary layer on a 3-D Body Started Impulsively from Rest"

23. A. Cler, "Prediction Theorique des Decollements sur Fuselages d'Helicopteres - Application aux Travaux d'Avant-Projet"

24. W. R. Van Dalsem and J. L. Steger, "Using the Boundary-Layer Equations in Three-Dimensional Viscous Flow Simulation"

25. J. C. Le Balleur and M. Lazareff, "Calcul d'Ecoulements Tridimenstionnels par Interaction Visqueux-non Visqueux Utilisant La Methode MZM"

26. M. C. P. Firmin, "Applications of RAE Viscous Flow Methods near Separation Boundaries for Three-Dimensional Wings in Transonic Flow"

27. J. S. Shang and W. L. Hankey, "Application of the Navier-Stokes Equations to Solve Aerodynamic Problems"

28. J. C. Wai, G. Blom, and H. Yoshihara, "Calculations for a Generic Fighter at Supersonic High Lift Conditions"

29. D. S. Chaussee, "High Speed Viscous Flow Calculations About Complex Configurations"

30. J. Flores, T. L. Holst, K. L. Gundy, U. Kaynak, and S. D. Thomas, "Transonic Navier-Stokes Wing Solution Using a Zonal Approach: Part I - Solution Methodology and Code Validation"

    N. Chanderjian
    "PART II - High Angle-of-Attack Simulation"

31. W. Kordulla, H. Vollmers and U. Dallmann, "Simulation of Three-Dimensional Transonic Flow with Separation Past a Hemisphere-Cylinder Configuration"

32. J. S. Shang and S. J. Scherr, "Numerical Simulation of the Flowfield around a Complete Aircraft"

APPENDIX B. Program Committee

Dr. R. E. WHITEHEAD (Co-Chairman)
Fluid Dynamics Program
Office of Naval Research
800 N. Quincy Street
Arlington, VA 22217
USA

M. l'Ing. General C. CAPELIER (Co-Chairman)
Directeur de l'Aerodynamique
B.P. 72
ONERA
92322 Chatillon
France

Dr. J. A. ESSERS
Universite de Liege
Institut de Mechanique
Service d'Aerodynamique Appliquee
Rue Ernest Solvay
B-4000 Liege
Belgium

Mr. L. H. OHMAN
Head, High Speed Aerodynamics Lab.
National Aeronautical Establishment
National Research Council - Montreal Road
Ottawa, Ontario K1A 0R6
Canada

Mr. l'Ing. en Chef B. MONNERIE
Chef de la Division d'Aerodynamique Appliquee
ONERA
B.P. 72
92322 Chatillon
France

Dipl. Ing. P. W. SACHER
Messerchmitt-Bolkow-Blohm-GmH
LKE 122
Postfach 80 11 60
D-8000 Munchen 80
Germany

Lt. Col. A. G. PANARAS, Dr. Ing.
HAF Technology Research Center (KETA)
Palaion Phaliron
Athens
Greece

Prof. Dr. L. G. NAPOLITANO, Ph.D.
Chair of Aerodynamics
Faculty of Engineering University of Naples
P. Le Tecchio 80
80125 Napoli
Italy

Mr. J. W. SLOOFF
National Aerospace Laboratory
Anthony Fokkerweg 2
1059 Amsterdam
Netherlands

Prof. Dr. H. NORSTRUD
Division of Hydro- and Gas Dynamics
The University of Trondheim
N-7034 Trondheim-NTH
Norway

Prof. Dr. C. CIRAY
Aeronautical Engineering Dept.
Middle East Technical University
Ankara
Turkey

Mr. D. H. PECKHAM
Superintendent AE1 Division
Royal Aircraft Establishment
R 141 Building
Farnborough
Hants GU14 6TD
United Kingdom

Dr. Leonard Roberts
Joint Institute for Aeronautics and Acoustics
Dept. Aeronautics and Astronautics
Stanford University
Stanford, CA 94305
USA

1. Report No.
NASA TM-88356

2. Government Accession No.

3. Recipient's Catalog No.

4. Title and Subtitle
AGARD FLUID DYNAMICS PANEL SYMPOSIUM ON APPLICATIONS OF COMPUTATIONAL FLUID DYNAMICS IN AERONAUTICS

5. Report Date
December 1986

6. Performing Organization Code

7. Author(s)
W. J. McCroskey

8. Performing Organization Report No.
A-86399

9. Performing Organization Name and Address
Ames Research Center, and
Aeroflightdynamics Directorate, U.S. Army Aviation
Research and Technology Activity, Ames Research
Center, Moffett Field, CA 94035

10. Work Unit No.

11. Contract or Grant No.

12. Sponsoring Agency Name and Address
National Aeronautics and Space Administration
Washington, DC 20546; and U.S. Army Aviation
Systems Command, St. Louis, MO 63120

13. Type of Report and Period Covered
Technical Memorandum

14. Sponsoring Agency Code
992-21-01-90

15. Supplementary Notes
Point of Contact: W. J. McCroskey, Ames Research Center, M/S 202A-1
Moffett Field, CA 94035 (415) 694-6428 or FTS 464-6428

16. Abstract

The Fluid Dynamics Panel of AGARD arranged a Symposium on "Applications of Computational Fluid Dynamics in Aeronautics," on 7-10 April 1986 in Aix-en-Provence, France. The purpose of the Symposium was to provide an assessment of the status of CFD in aerodynamic design and analysis, with an emphasis on emerging applications of advanced computational techniques to complex configurations. Sessions were devoted specifically to grid generation, methods for inviscid flows, calculations of viscous-inviscid interactions, and methods for solving the Navier-Stokes equations. The 31 papers presented at the meeting are published in AGARD Conference Proceedings CP-412 and are listed in the Appendix of this report. A brief synopsis of each paper and some general conclusions and recommendations are given in this evaluation report.

17. Key Words (Suggested by Author(s))
Computational Fluid Dynamics
Three-dimensional flows

18. Distribution Statement
Unclassified-Unlimited

Subject Category - 02

19. Security Classif. (of this report)
Unclassified

20. Security Classif. (of this page)
Unclassified

21. No. of Pages
18

22. Price*
A02

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