NASA Technical Memorandum 88246

NASA TM - 88246

NASA-TM-88246 19860018585

A Perspective of Computational Fluid Dynamics

Paul Kutler

May 1986

NF00952

NASA

National Aeronautics and
Space Administration

$\mathcal{N}86\text{-}28057$ #

A Perspective of Computational Fluid Dynamics

Paul Kutler, Ames Research Center, Moffett Field, California

May 1986

NASA

$\mathcal{N}86\text{-}28057$ #

A PERSPECTIVE OF COMPUTATIONAL FLUID DYNAMICS

Paul Kutler
NASA Ames Research Center
Moffett Field, California 94035, U.S.A.

SUMMARY

Computational fluid dynamics (CFD) is maturing, and is at a stage in its technological life cycle in which it is now routinely applied to some rather complicated problems; it is starting to create an impact on the design cycle of aerospace flight vehicles and their components. CFD is also being used to better understand the fluid physics of flows heretofore not understood, such as three-dimensional separation. CFD is also being used to complement and is being complemented by experiments. In this paper, the primary and secondary pacing items that governed CFD in the past are reviewed and updated. The future prospects of CFD are explored which will offer people working in the discipline challenges that should extend the technological life cycle to further increase the capabilities of a proven and demonstrated technology.

1. INTRODUCTION

Fifteen years ago computation fluid dynamics (CFD) was in its infancy, and on the first part of its technological life cycle curve (Fig. 1). The facets that comprise the discipline of CFD, such as algorithm development, grid generation, geometry definition, boundary and initial conditions, turbulence modeling, pre- and post-data processing, computer technology, etc., were all ripe for technological advances. Many simple problems, easily amenable for solution using CFD, were unsolved. At the same time, there were very few researchers working in the discipline and on its various facets (Fig. 2). Also, very little computer power was devoted to, or available for, solving CFD problems. Because of the demonstrated potential of CFD, and the understood limitations of experimental testing (see Chapman, Mark, and Pirtle [1]), the discipline of CFD developed at a rapid pace. Today, because of the manpower and computer resources devoted to CFD, technological advances in the discipline have matured to the point at which CFD is
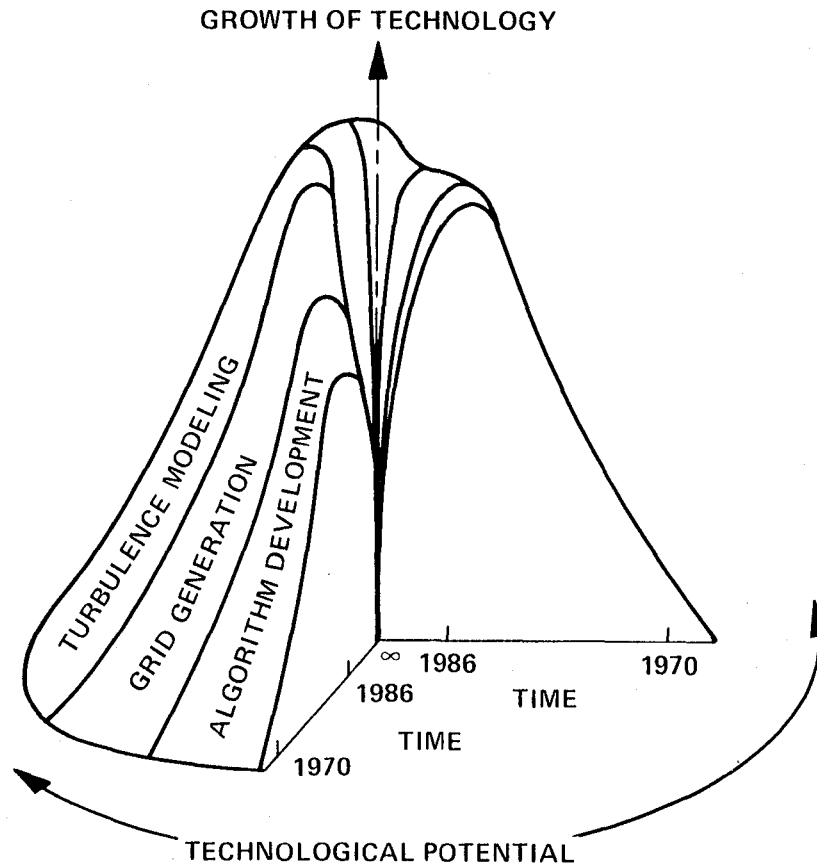
1

Figure 1. Rate of growth of CFD technology.

becoming routine, and thus near the growth peak of its tech-
nological life-cycle curve. Most of the simple problems have
been solved, and only the very complex and difficult ones
remain. To extend the life-cycle curve of CFD, it will be
necessary, for example, to seek new disciplines that can be
coupled with CFD, to utilize CFD for understanding or discov-
ering new fluid-flow phenomena, or to apply CFD to new aero-
nautical challenges offered by future aerospace vehicles.

In the future, aerospace manufacturers will rely exten-
sively on numerical simulations because of 1) the demonstrated
capability of CFD, 2) the increasing number of supercomputers
available for performing CFD simulations, 3) the lack of
ground-based experimental facilities in the flow regimes of
interest, and 4) the saturation of existing experimental
facilities. Instead of simply utilizing national laboratories
for their unique experimental facilities, vehicle designers
will be able to perform corresponding numerical simulations on
national computational facilities, thereby complementing their
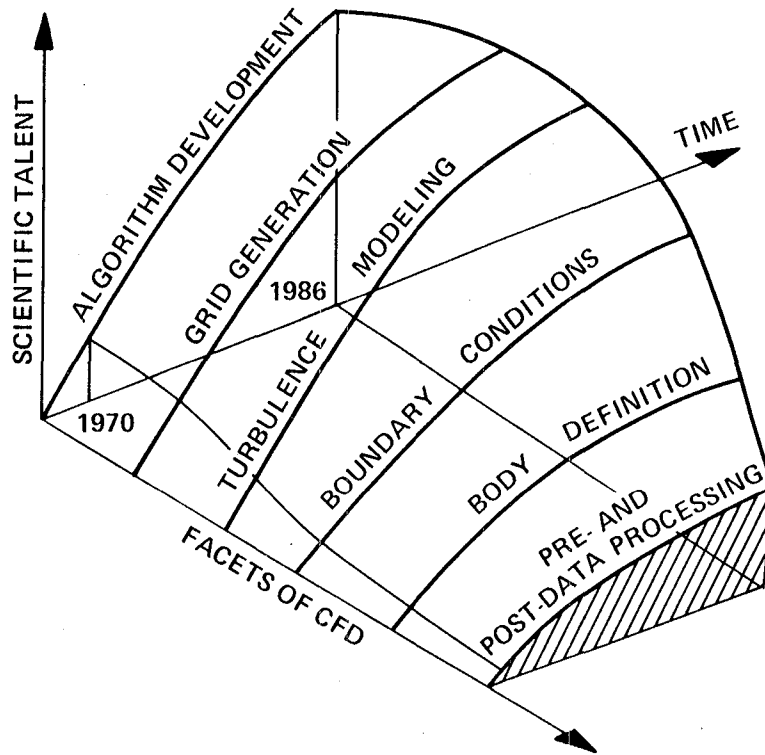experimental test programs.

Figure 2. Scientific talent devoted to facets of CFD as a function of time.

2. STATUS OF PRIMARY AND SECONDARY PACING ITEMS

Chapman [2], in 1981, outlined pacing items for CFD that included three-dimensional (3D) grid generation, turbulence modeling, algorithm development, and computer-mainframe design advances. In a 1983 AIAA Paper (later published in the AIAA Journal), Kutler [3] updated the list and classified the pacing items according to primary and secondary items. The primary items included grid generation, turbulence modeling, computer power, and solution methodologies; the secondary items included algorithm development, complex-geometry definition, and pre- and post-data processing. In this section, the pacing items of the past are reviewed and updated, and the major accomplishments are summarized.

2.1 Computer technology

The continued demand for more powerful computers by computational fluid dynamicists to perform not only direct Navier-Stokes simulations for simple geometries, but Reynolds-averaged Navier-Stokes (RANS) calculations for complex 3D configurations, continues to pace the progress of CFD. Also, the problems of the future (e.g., in hypersonics) that require the solution not only of the Navier-Stokes equations, but also

3

of the finite-rate chemistry equations, will require a computer that operates orders of magnitude faster than those currently available (Fig. 3).

The balance between the speed and memory of a given computer is essential for the effective use of the machine. Too much memory without adequate speed, or too little memory with an abundance of speed, is inefficient. A summary of the existing and planned scientific supercomputing systems is shown in Table 1. It is, however, safe to say that, regardless of the power offered by the latest state-of-the-art computer, the computational fluid dynamicists will easily saturate it and demand more power.

A simple advantage that computers/software have over wind tunnels in performing simulations is that they can be continually and inexpensively upgraded to enhance their capabilities over time whereas the wind tunnels cannot. Computing centers such as the Numerical Aerodynamic Simulation (NAS) Facility, located at NASA Ames, and the National Science Foundation (NSF) centers, distributed around the United States, are cheaper to build and to upgrade than are new wind tunnel facilities. Computational centers, and the simulations performed with them, can help eliminate the saturation of existing wind tunnels, and thus make the use of these experimental facilities more effective and efficient.
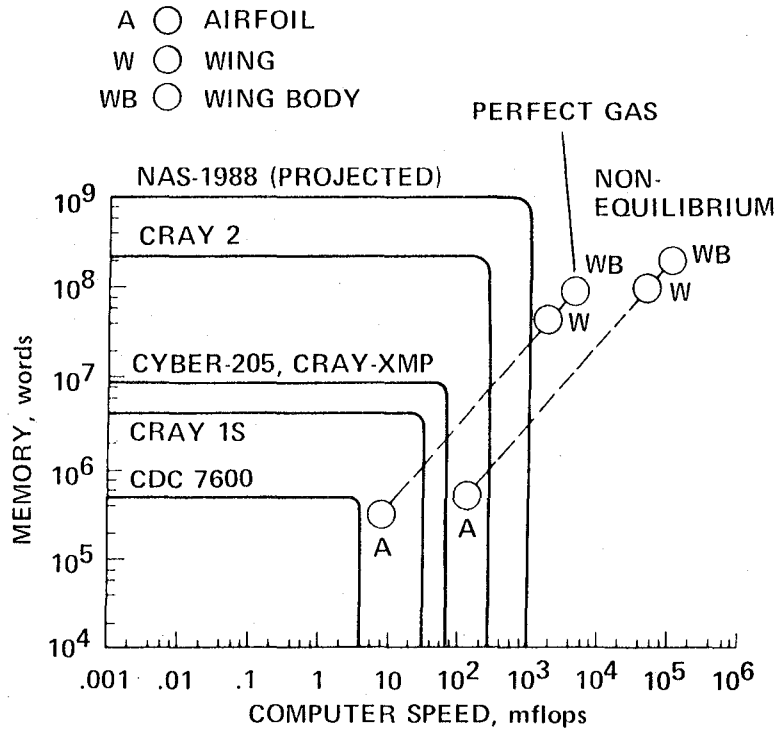
Figure 3. Computer speed and memory requirements (15-min runs, 1985 algorithms).

4

SYSTEM	CLOCK CYCLE, nsec	MAIN MEMORY SIZE, megabytes/64-bit megawords	SECONDARY MEMORY SIZE,	MAXIMUM/ SUSTAINED VECTOR RATE, mflops[a]	NO. OF PROCESSORS	NO. OF PIPES/PROC.	DATE AVAILABLE
CRAY XMP	9.5	128/16	1024/128	760[b]/400[c]	4	1	85
CRAY 2 (C2)	4.1	2048/256	—	1620[b]/400[c]	4	1	2Q 85
CRAY YMP	NA[d]	NA	NA	3XC2[e]	NA	NA	2Q 87
CRAY 3	NA	NA	NA	10XC2[e]	NA	NA	1Q 88
ETA 10	7.0	256/32	2048/256	5000/2000[c]	NA	2	3Q 86
AMDAHL 1400	7.5	256/32	—	1133/NA	1	4	4Q 84
HITACHI S820/20	14	256/32	1024/128	630/NA	1	1	3Q 84
NEC SX-2	6	256/32	2048/256	1300/NA	1	4	1Q 86

[a]DATA FOR FULL 64-BIT PRECISION
[b]DEMONSTRATED PERFORMANCE
[c]ESTIMATE FOR OPTIMIZED CFD CODE
[d]INFORMATION NOT AVAILABLE
[e]ESTIMATED PERFORMANCE IMPROVEMENT OVER CRAY 2

Table 1. Characteristics of existing and planned computer systems.

5

2.2 Turbulence physics and modeling

The simulation of viscous flows is being attacked from both ends of the computational spectrum. At one end are the scientists who solve the RANS equations with a suitable turbulence model derived from theoretical analysis and data obtained from building-block experiments. At the other end of the spectrum are the scientists who solve the complete Navier-Stokes equations with either a sub-grid model for the small scales of turbulence (large eddy simulation), or no model at all (direct simulation). As time progresses and CFD and computer technology mature, the two approaches will merge.

The development of suitable turbulence models for the RANS equations to date remains highly problem-dependent. According to Marvin [4], to improve the turbulence-model development process, it should be considered at the early stage of code development. The first step in such a process is to identify those flows pacing the development of the aerodynamic computations. The second step is to develop models through a phased approach of building-block studies that combine theory, experiment (requiring new instrumentation for extracting refined data), and computations. The final step is to provide verification and/or limits of the modeling through benchmark experiments over a practical range of Reynolds and Mach numbers.

The application of different turbulence models in different regions of the flow (e.g., in a zonal approach), might be required to obtain better accuracy of the simulation process. Also, as the Mach number of the free-stream flow increases, the incompressible turbulence models developed to date will probably not work. Thus it will be necessary to develop new models, including previously neglected Reynolds stress terms, for treating these flows.

As the direct numerical simulations progress, it is becoming possible to utilize the numerical data generated from those solutions to create turbulence models. Such data provides much more information than can be gleaned from an experiment, thus permitting the construction of better turbulence models. The limitation is with the simplified and low Reynold's number flows attainable by direct simulations.

2.3 Solution methodology development

Two noteworthy solution methods that have resulted in advancements of the state of the art in CFD include a zonal procedure developed by Rai [5] for treating a rotor-stator combination (i.e., a multiple moving-body problem), and a tetrahedron procedure developed by Jameson [6] for treating commercial aircraft configurations. Rai's procedure is applicable for moving bodies and required the development of

boundary condition procedures for transmitting information from one moving grid to the other.

Jameson's procedure permits the easy treatment of complicated configurations because the flow region of interest is discretized using tetrahedrons. He developed a boundary-condition procedure for conserving fluxes across the volume surfaces that did not degrade the accuracy of the solution. He also developed a data-management system for handling the randomly ordered control volumes.

The results from both of these procedures are formidable. A typical viscous-flow result for the rotor stator is shown in Fig. 4, for blunt-nose blades. The instantaneous velocity vectors, with the free stream subtracted out, are displayed, clearly showing the vortex pattern behind the blades.

To facilitate the use of one's computer program by aerospace vehicle designers, code developers should strive to make the code robust. In essence, this means that, with very few limitations, the program should be capable of yielding results of which the accuracy is predictable, and with little intervention from the user. It should have few flow restrictions, except for the limits of applicability of the governing equation set, and should be capable of treating complex and varied geometries.
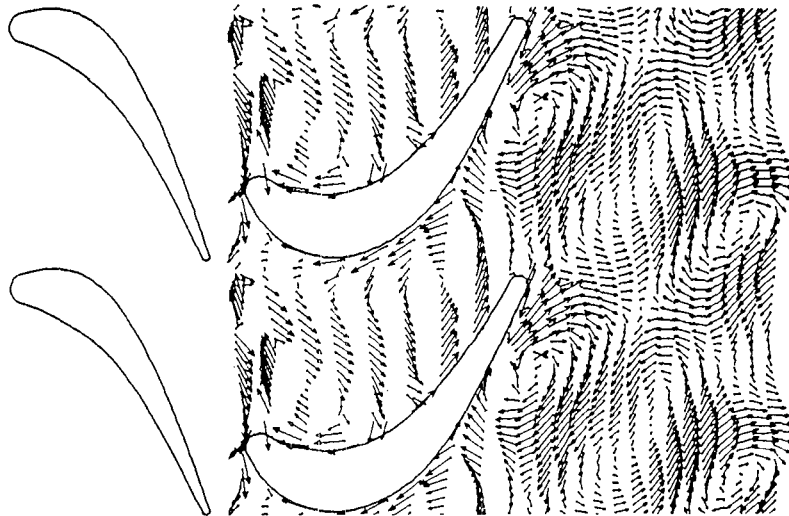

Figure 4. Instantaneous-velocity vector for rotor-stator configuration.

2.4 Algorithm development

Considerable human talent has been devoted to the development of more accurate and faster algorithms for solving the gas-dynamic equations, and has resulted in considerable progress. There is, however, still more progress to be made in this facet of CFD. The current set of equations being attacked by algorithm developers is the Navier-Stokes (including both complete and Reynold's-averaged). For unsteady solutions of these equations, existing algorithms are capable of efficiently obtaining solutions; however, more improvement is still possible in developing algorithms for obtaining convergence to the steady state.

Also, to devise schemes for faster convergence, developers are striving for more accurate and robust algorithms, particularly in the area of shock-capturing properties. They are trying to minimize the number of free parameters that users must select to utilize an algorithm optimally. Total variational diminishing (TVD) and upwinding schemes satisfy this criterion. On a per-point basis, however, such schemes require more computer time, but the results they yield are worth this time. Figure 5 shows the TVD results of Yee [7] for a planar blast wave passing over an airfoil. These results clearly depict the intricate wave pattern of the flow, including the slip surface generated from the triple point near the trailing edge of the airfoil. This solution was obtained without the use of a solution-adaptive grid, but with such a grid, the results could be enhanced even further.
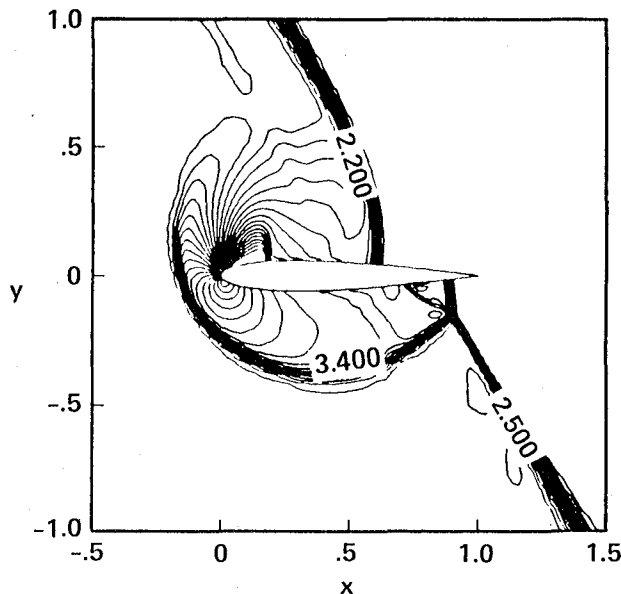

Figure 5. Density contours for blast wave striking an airfoil.

8

The supercomputers now available are causing some researchers to explore old procedures for solving the Navier-Stokes equations that some time ago seemed impractical because of computer limitations. Beam and Bailey [8] are looking at Newton's method, which does not involve any approximation, as do most conventional methods. Such a procedure involves the inversion of a rather large matrix, but could be used to evaluate the effects of approximations made in other procedures.

2.5 Geometry definition

Considerable progress has been made in the geometry-definition facet of CFD (i.e., translating a complicated configuration into data understandable by the flow solver), but it is still not a routine process. As the sophistication of computer programs increases for treating complicated configurations, and their routine use by designers also increases, the demand for easily applied geometry-definition procedures will also increase. To date, flows about configurations such as commercial aircraft [6], fighter planes [9], spacecraft, and complicated components of those vehicles [10] are being computed and hence require such a geometry tool. Figure 6 is typical of the complex configurations being studied today. The amount of time required to input the geometry describing configurations like these into the computer, however, is on the order of several months.

It is important that this process becomes routine (i.e, require only a few hours, and possibly only minutes), and thus not slow the progress of CFD. This will require the development of sophisticated software. Such software should be designer-friendly for versatile use and employ high-level computer graphics.
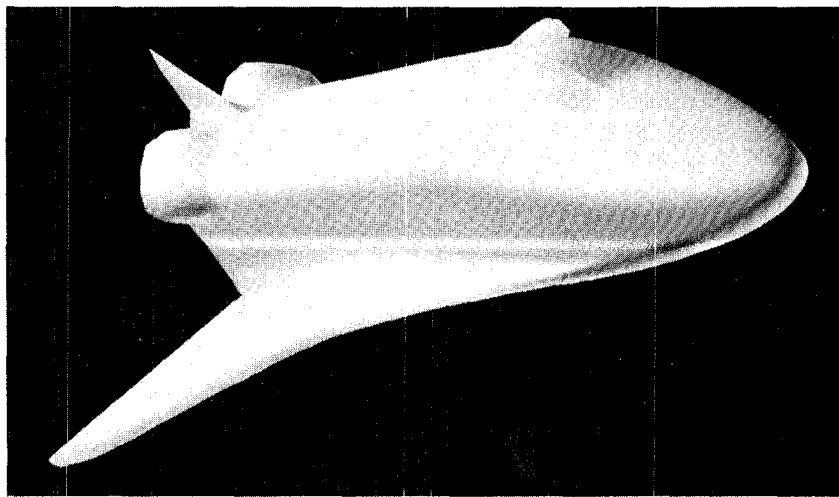

Figure 6. Computer-generated geometry for Space Shuttle.

9

2.6 Grid generation

The process of grid generation has received considerable
attention by scientists because of the need to efficiently and
effectively distribute grid points to generate the most accu-
rate solution possible. To date, complicated configurations
have been treated computationally. Examples are shown in
Figs. 7 and 8, in which different grid topologies have been
used to discretize the flow about the Space Shuttle (a single
module grid) and a generic aircraft (a multiple module or
block grid). The grid for the Shuttle used a hyperbolic
solver, whereas the grid for the aircraft used an elliptic
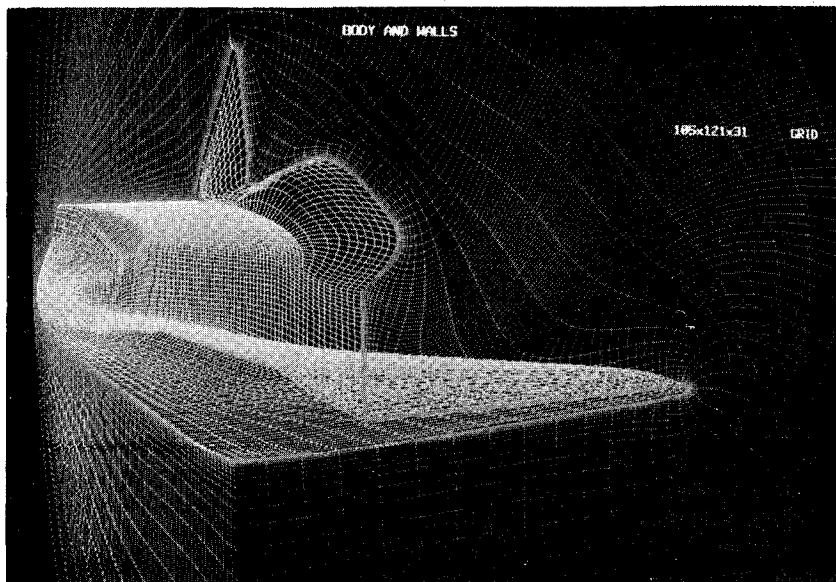solver.

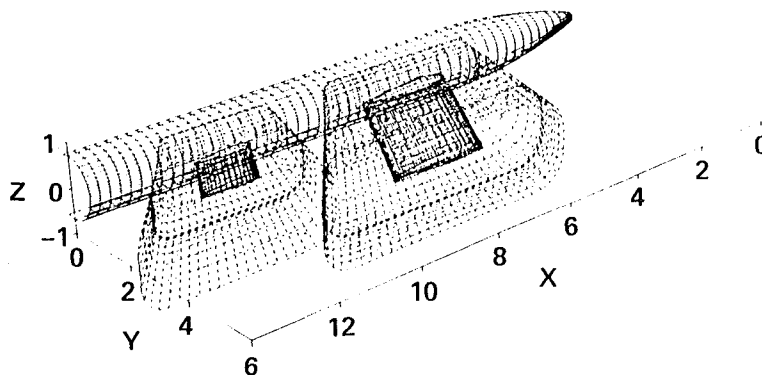Figure 7. Single module grid for Space Shuttle.

Figure 8. 3D wing/body/tail overset grid configuration.

Because of the CFD scientist's desire for accuracy and efficiency, solution-adapative grid procedures have gained popularity. Typical of the solution enhancements possible using such procedures are the results of Nakahashi and Deiwert [11], as shown in Fig. 9 for a two-dimensional airfoil.

Unsteady problems involving the motion of one body relative to another have created another set of grid-generation problems. An example of such a problem is the store drop from an aircraft. To treat this problem, a component-adaptive, overlapping grid system is used. A typical grid generated by Dougherty [12] for this problem is shown in Fig. 10.

The automation of the grid-generation procedures to facilitate discretizing the flows about complicated configurations should be of paramount importance to those working in this facet of CFD. This is an area in which expert systems might help to eliminate the need for routine application of conventional grid-generation procedures.

2.7 Pre- and post-data processing

With the placement of supercomputers around the world, and the use of these machines for solving complicated 3D problems, comes the necessity for managing enormous amounts of data. To accomplish this, the user community has relied
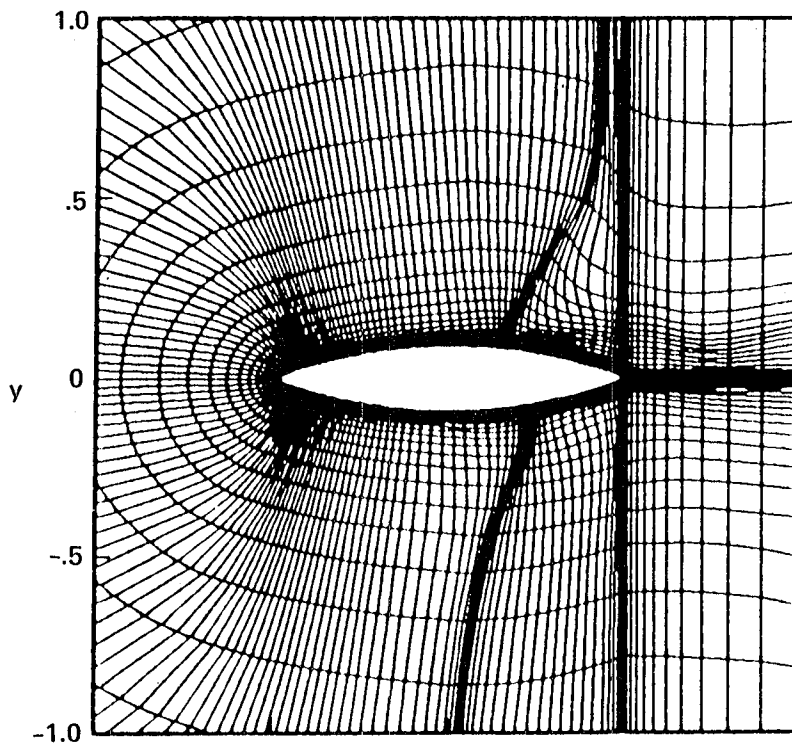
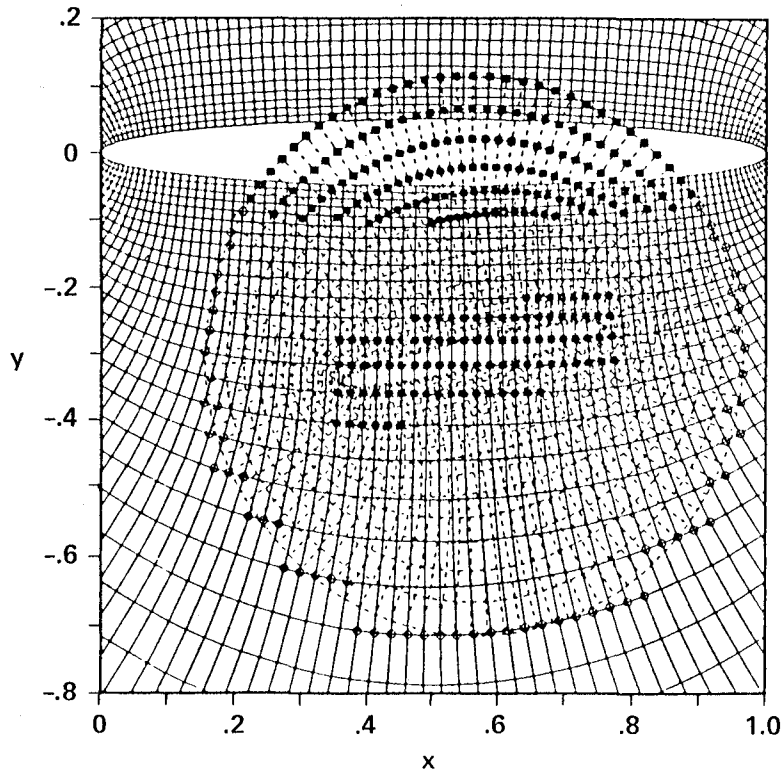Figure 9. Solution-adapative grid for airfoil with buffet.

Figure 10. Overset grid for store-separation simulation.

heavily on high-resolution, high-throughput computer-graphics devices.

Sophisticated software packages (e.g., that by Buning [13]), have been designed which permit the viewing of the results in either static or dynamic motion for the analysis and understanding of the flow-field data. Such software packages, however, are passive in the sense that they display only what they are told to display. In such a process, it is possible that some of the interesting or undiscovered fluid physics might be masked. Therefore, what is needed is an active or "smart" software display package that searches the data base for interesting flow phenomena and displays them. This would require, for example, the program to look for different combinations of the flow variables and their gradients, or derivatives of their gradients, to uncover interesting regions of the flow that might be lurking in the small-scale portions of the mesh, such as secondary, separated flow regions, and call the viewers attention to it and display it.

3. FUTURE PROSPECTS FOR CFD

Numerical simulations can now be performed on many complex configurations. The capability of CFD is such that many complicated problems can be solved if the resources are

12

channeled into the effort. Scientists with the freedom to select their computational research tasks are now faced with the decision as to which problems to solve. To aid in their decision, certain criteria might be suggested, such as 1) is the problem of national importance, 2) will its solution lead to a new design tool, 3) will it aid in the understanding of complex fluid physics or the discovery of new flow phenomena, 4) will it push the state of the art in computational fluid dynamics, and 5) is the problem tractable in a finite amount of time.

The design and construction of future CFD-applications software is becoming continually more complicated because of the complex problems being addressed, requiring teams of researchers. Because of the complexity of the codes, it is critical that the eventual user be involved in the software development stage. This requires that the code builders get "close to the customers." The customers can make constructive suggestions in the program's design, can familiarize themselves with the program, and will thus be more willing to use it when it is completed. Because of the involvement of the customer, the resulting code will be "designer-friendly." Subsequent use of the code in the vehicle-design process, however, will depend on the confidence level designers have of the code and the predictable accuracy of the results generated by the code. That confidence level is enhanced by involving the vehicle designer in the program's development.

Because of the complexity of CFD software, it is vital that the proper program documentation exists. It should not be the duty of the research scientist who conceived the program to provide the documentation (although it should be his or her responsibility), but rather a programmer well versed in such duties who works with the scientist in the development process. The understanding and use of complex codes can also be taught by the computer. Expert systems can easily be constructed to train potential users how to efficiently utilize complex CFD software.

A considerable number of challenging technical areas exist for which CFD will be beneficial, and sometimes mandatory. Two areas of particular importance that will be addressed here are unsteady flows and interdisciplinary physics.

Unsteady-flow problems result from the instability of separated flow regions, or from the relative motion of one body with respect to another. Two problems in which relative body motion causes unsteady flow are the helicopter and rotating turbomachinery. Both problems satisfy most of the criteria outlined at the beginning of this section. To date, neither problem has been simulated computationally (only components of

13

the problem), but formidable efforts are under way that will lead to their eventual numerical simulation.

Researchers such as Davis and Chang [14], and McCroskey and Bader [15], are developing Euler or RANS programs for computing the unsteady flow about helicopter rotors. Unsteady problems such as the blade-vortex interaction have been simulated using these codes (Fig. 11). Inclusion of the helicopter fuselage and tail rotor in these calculations poses considerable challenge for the scientists, but the problem is not technology-limited, and will eventually be simulated using CFD. It will thus lead to a better understanding of helicopter aerodynamics and improved designs.

The rotor-stator problem in two dimensions has been successfully simulated by Rai [5]. The solution of this problem required advancements in the state of the art in CFD regarding boundary conditions and grid generation. Extension of this technology to three dimensions is formidable but tractable. Simulation of the 3D problem could yield solutions for flows through propellers, pumps, compressors, and turbines, and eventually lead to the development of more efficient propulsors and jet engines.

The simulation of unsteady viscous flows about realistic aircraft configurations is now possible using computational tools. Several computer codes for simulating these flows have been developed by various researchers throughout the United States. With these codes, it will now be possible to begin studying unsteady flow problems that result when high-performance aircraft/spacecraft fly at large angles of attack. These unsteady flows include assymetric vortex shedding and vortex breakdown or bursting. It will also be possible to use these codes for predicting aircraft/spacecraft performance near their performance boundaries. Numerical results by Rizk [16] for flow over the Space Shuttle at Mach 1.4 and 0° angle of attack, are shown in Fig. 12, and demonstrate the capability of today's technology.

Interdisciplinary physics will not only offer significant challenges to the research scientist, but also challenge the power of existing or planned computational facilities. The mating of different technical disciplines into one computer program for more relevant design applications should tax both the scientist and the machine. In the past, simple couplings have occurred, such as linking a structural response code to a fluid dynamics code to study aeroelastic problems (see Goorjian, et al. [17]), or the coupling of a flow code with an optimization routine for wing design (see Cosentino [18]). In the future, other disciplines will be linked with flow codes such as propulsion and controls.
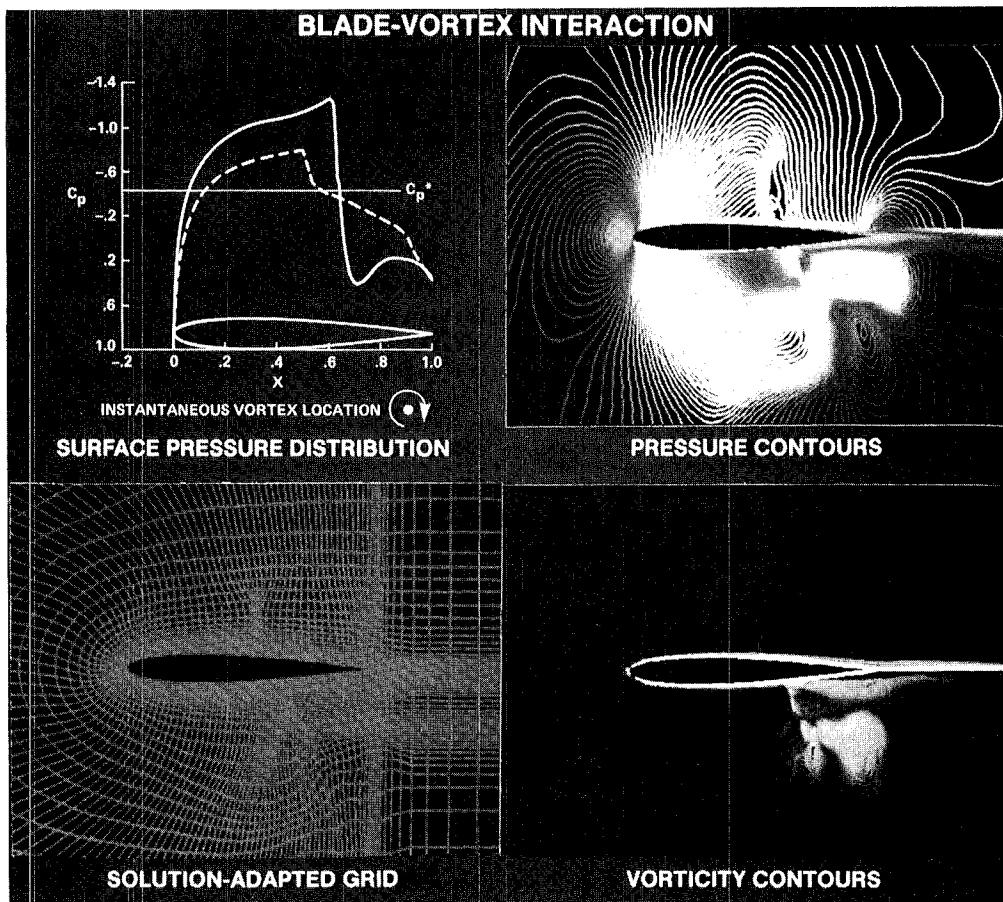
14

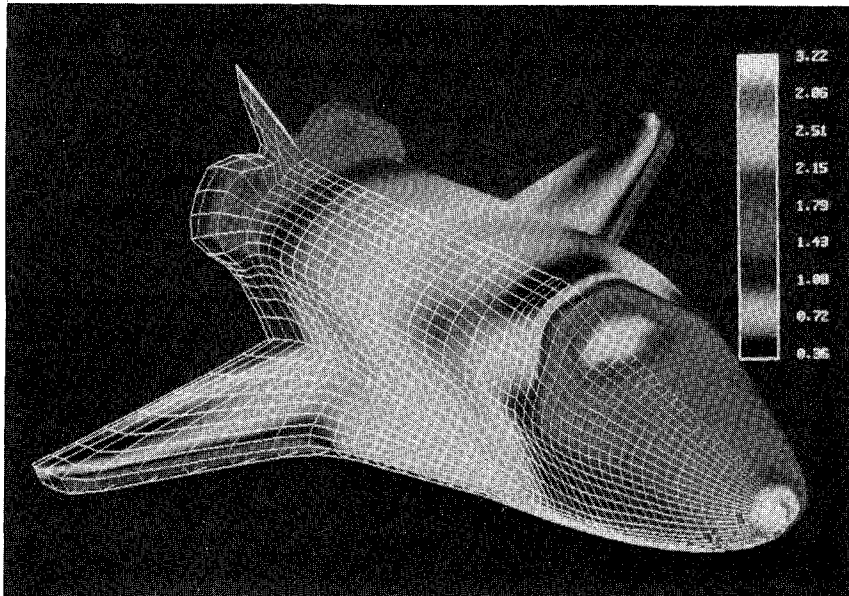Figure 11. Two-dimensional blade-vortex interaction.

Figure 12. Pressure contours from viscous solution for Space Shuttle.

As the need grows for high-speed flight, the gas-dynamic equations routinely solved today governing those flows will increase in complexity because of strong shocks and thermal and chemical nonequilibrium phenomena. Fluid dynamicists and chemists will begin working together to couple their disciplines and study problems involving dissociation and ionization, reaction rates, radiation physics, and equilibrium constants. Efficient algorithms for computing such flows will have to be invented to treat "stiff" equations. These inderdisciplinary equation sets will challenge algorithm developers in the future.

Turbulence models, based on compressible flow theory, will have to be developed because the existing models, based on incompressible flow, break down for high-speed flight. In addition, it will be possible to begin using numerically generated data (i.e., from direct simulations) to extract data required for the development of turbulence models. This is happening on a limited basis for incompressible flows. CFD will play an important role in the high-speed flight regime because of the lack of ground-based experimental facilities.

The introduction of high-speed flight and its associated problems to the CFD community will serve to extend the rate of growth of the technological life cycle of the discipline. This will offer scientists working in the various facets of CFD opportunities and challenges similar to those that existed over the last decade for transonics.

It is important that in the future fluid dynamicists, whether they use computational or experimental tools for their trade, work closely together. The synergy to be garnered is too valuable not to take advantage of such a cooperative arrangement. Computer codes require validation experiments, and experiments require supplemental computations. Research laboratories, whose basic product is information, and which possess the facilities for experimental, computational, and flight testing, will offer the greatest possiblity for synergy, and will produce the most valuable technical product.

4. CONCLUSIONS

Computers are assuming an increasingly important role in aerospace research and development. Considerable resources (manpower and computer) have been channeled into CFD. As a result, CFD is rapidly becoming an extremely powerful tool in the design process, as well as in the understanding of complex fluid physics. Substantial payoffs in both areas have been demonstrated. On the other hand, the rate of technological growth in CFD is naturally decreasing because of the substantial resources devoted to its development. It is therefore mandatory that new challenges be offered to the CFD research scientist to extend the life cycle rate of growth of CFD technology. Unsteady flow problems and interdisciplinary physics can serve this purpose. Supercomputing centers such as NAS, located at NASA Ames Research Center, and the NSF Centers, located throughout the United States, will play a key role in advancing the state of the art in CFD, and will be a critical element in the national base of aeronautical facilities.

Lack of uniqueness, or the narrowing gap, in computing facilities between foreign countries and the United States, plus lack of uniqueness in CFD technology and talent, make CFD competitive on an international level. This intensity of global competition in "high tech" areas such as CFD could pose a serious challenge to the aeronautical preeminence of the United States.

REFERENCES

1. CHAPMAN, D.R., MARK, H., and PIRTLE, M.W. - 'Computers vs. Wind Tunnels for Aerodynamic Flow Simulations'. Astronaut. & Aeronaut., 1975, 13, 22-35.

2. CHAPMAN, D.R. - 'Trends and Pacing Items in Computational Aerodynamics'. Lecture Notes in Physics, 1981, 141, Ed. Reynolds, W.C. and MacCormack, R.W., Springer-Verlag, Berlin Heidelberg, Germany.

3. KUTLER, P. - 'A Perspective of Theoretical and Applied Computational Fluid Dynamics'. AIAA Journal, 1985, 23.

4. MARVIN, J.G. - 'Future Requirements of Wind Tunnels for Computational Fluid Dynamics Code Verifications'. AIAA Paper 86-0752-CP, 1986.

5. RAI, M.M. - 'Navier-Stokes Simulations of Rotor-Stator Interaction Using Patched and Overlaid Grids'. AIAA Paper 85-1519, 1985.

6. JAMESON, A., BAKER, T.J., and WEATHERILL, N.P. - 'Calculation of Inviscid Transonic Flow Over a Complete Aircraft'. AIAA Paper 86-0103, 1986.

7. YEE, H. - 'Numerical Experiments with a Symmetric High-Resolution Shock-Capturing Scheme'. Proc. 10th Intl. Conf. on Num. Meth. in Fluid Dynamics, June 1986, Beijing, China.

8. BEAM, R.M. and BAILEY, H.E. - pvt. communications, January 1986.

9. EDWARDS, T.A. - 'Definition and Verification of a Complex Aircraft for Aerodynamic Calculations'. AIAA Paper 86-0431, 1986.

10. KWAK, D., CHANG, J.L.C., and SHANKS, S.P. - 'A Solution Procedure for Three-Dimensional Incompressible Navier-Stokes Equation and Its Application'. Proc. 9th Int. Conf. on Num. Meth. in FLuid Dynamics, 1984.

11. NAKAHASHI, K. and DEIWERT, G.S. - 'A Self-Adaptive-Grid Method with Application to Airfoil Flow'. AIAA Paper 85-1525, 1986.

12. DOUGHERTY, F.C., BENEK, J.A., and STEGER, J.L. - 'On Applications of Chimera Grid Schemes to Store Separation'. NASA TM 88193, 1985.

13. BUNING, P.G. and STEGER, J.L. - 'Graphics and Flow Visualization in Computational Fluid Dynamics'. AIAA Paper 85-1507, 1985.

14. DAVIS, S.S. and CHANG, I.-C. - 'The Critical Role of Computational Fluid Dynamics in Rotary-Wing Aerodynamics'. AIAA Paper 86-0336, 1986.

15. McCROSKEY, W.J. and BAEDER, J.D. - 'Some Recent Advances in Computational Aerodynamics for Helicopter Applications'. NASA TM 86777, 1985.

16. RIZK, Y.M. and BEN-SHMUEL, S. - 'Computation of the Viscous Flow Around the Shuttle Orbiter at Low Supersonic Speeds'. AIAA Paper 85-0168, 1985.

17. GURUSWAMY, G.P., GOORJIAN, P.M., and TU, E.L. - 'Unsteady
     Transonics of a Wing with Tip Store'. AIAA Paper 86-0010,
     1986.

18. COSENTINO, G. and HOLST, T.L. - 'Numerical Optimization
     Design of Advanced Transonic Wing Configurations'. AIAA
     Paper 85-0424, 1985.

1. Report No. NASA TM 88246	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle A PERSPECTIVE OF COMPUTATIONAL FLUID DYNAMICS		5. Report Date May 1986
		6. Performing Organization Code
7. Author(s) Paul Kutler		8. Performing Organization Report No. A-86224
		10. Work Unit No.
9. Performing Organization Name and Address Ames Research Center Moffett Field, CA 94035		11. Contract or Grant No.
		13. Type of Report and Period Covered Technical Memorandum
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546		14. Sponsoring Agency Code 505-60-01

16. Abstract

Computational fluid dynamics (CFD) is maturing, and is at a stage in its technological life cycle in which it is now routinely applied to some rather complicated problems; it is starting to create an impact on the design cycle of aerospace flight vehicles and their components. CFD is also being used to better understand the fluid physics of flows heretofore not understood, such as three-dimensional separation. CFD is also being used to complement and is being complemented by experiments. In this paper, the primary and secondary pacing items that governed CFD in the past are reviewed and updated. The future prospects of CFD are explored which will offer people working in the discipline challenges that should extend the technological life cycle to further increase the capabilities of a proven and demonstrated technology.

17. Key Words (Suggested by Author(s)) Computational fluid dynamics Aerodynamics Mathematics Computer science	18. Distribution Statement Unlimited Subject Category - 02
19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price*
Unclassified	Unclassified	21	A02

End of Document