N87-26005

CFD APPLICATIONS: THE LOCKHEED PERSPECTIVE

Luis R. Miranda
Lockheed-California Company

INTRODUCTION

The Numerical Aerodynamic Simulator (NAS) epitomizes the coming of age of supercomputing and opens exciting new horizons in the world of numerical simulation. The technological benefits likely to result from the use of the NAS may surprise even its most optimistic advocates. All this makes me both pleased and honored to be a participant of this conference which celebrates the inauguration of the NAS.

In this article I will give an overview of supercomputing at Lockheed Corporation in the area of Computational Fluid Dynamics (CFD). This overview will focus on developments and applications of CFD as an aircraft design tool and will attempt to present an assessment, within this context, of the state-of-the-art in CFD methodology. Although CFD is being actively developed and applied throughout Lockheed Corporation, the material of this paper draws heavily from activities that have taken place or are underway at Lockheed-California Company.

Of course, supercomputing in the aerospace industry is not limited to CFD. Many other disciplines either are already, or have the potential of, benefiting from supercomputing: structural analysis and optimization, vehicle signature (electromagnetic, acoustic, thermal) prediction, and signal processing, to name the obvious ones. In other areas, where the supercomputing revolution has not yet made its full impact, the payoffs are likely to be extraordinary. One such area is manufacturing. Supercomputers will make possible the effective integration of all phases and aspects of complex manufacturing processes and are bound to become essential in the development and operation of a new generation of sophisticated, versatile, and "intelligent" robotic systems.

In spite of the wide applicability of supercomputing, I have chosen to limit my discussion to CFD and its application to aircraft design because of the following factors: 1) time and space constraints; 2) personal bias due to my familiarity with the subject; and 3) CFD was one of the first disciplines to push the limits of "ordinary" or "general purpose" computers, thus giving rise to the need for supercomputer capability. Moreover, the practical feasibility of CFD application demands supercomputing power, and most future developments in this area hinge on the availability of even greater computer speed and memory capability.

SUPERCOMPUTERS AT LOCKHEED CORPORATION

The term supercomputer is a relative one; what we consider supercomputer performance today may become "ordinary" or "general purpose" computer capability in a few years. For the purpose of the present discussion, the term "supercomputer" will denote a machine with a performance capability of at least class VI in accordance with the following classification defined by the U.S. Department of Energy:

Class III	Sustained operating speeds between 0.6 and 2 million floating point operations per second (megaflops)
Class IV	2 to 6 megaflops
Class V	6 to 20 megaflops
Class VI	20 to 60 megaflops
Class VII	Greater than 60 megaflops

Presently two supercomputers are in operation at Lockheed Corporation. The first one, a Cray 1S, was installed at the Sunnyvale, California, plant of Lockheed Missiles and Space Company (LMSC) in 1982. This supercomputer is basically dedicated to serving the needs of the various LMSC divisions. The second one, a Cray X-MP/24, became operational at the Kelly Johnson Research Center, Rye Canyon, California, in April 1986.

The Cray X-MP/24 is a dual processor machine with an in-core memory capacity of 4 million 64-bit words. It is front-ended by two Digital Equipment Corporation (DEC) computers in the Lockheed installation: a DEC 8600 for unclassified processing, and a DEC 8300 for classified mode operation. This supercomputer, although it can be accessed remotely from all major Lockheed divisions, is primarily intended to cater to the Lockheed Aeronautical Systems Group companies.

The principal motivation for the acquisition of supercomputer hardware was the perceived need for processing speed. The above supercomputers have been able to fulfill that need fairly well for most present applications. At the same time, they have offered the possibility of more advanced computations. As we gain

more experience with these computations, and as we attempt to tackle more demanding problems, the need for more computer memory becomes acutely evident. This is particularly true in the engineering applications of CFD. As will be indicated later on, the use of advanced CFD methods in aircraft design applications requires fast-access memory capacity ranging from 16 megawords upwards.

THE CHALLENGE OF DESIGN APPLICATIONS

The paper is mainly concerned with CFD developments and applications within the context of aircraft design. Therefore, although it may appear obvious to many, it is important to emphasize the principal requirements that a CFD code ought to meet if it is to be used as an aircraft design tool, namely:

1) It must be a predictive tool, i.e., the accuracy and reliability of the computation should not depend on too much fine tuning or tailoring of input parameters. In many advanced applications, applicable experimental data are not available to "calibrate" the results of numerical computations.

2) It must be able to handle complex geometric configurations, including three-dimensionality, intricate aircraft components, and complete aircraft arrangements.

3) It must yield useful results in a reasonably short period of time, consistent with usually pressing program schedules.

4) Its application, although it may require a reasonable degree of expertise and experience, must not demand specialization to a degree that only its developers can effectively use it.

The above requirements are extremely important if CFD is to realize its full potential as an effective design tool. As has already been pointed out in Reference 1, these requirements directly relate to the equation:

EFFECTIVENESS = QUALITY X ACCEPTANCE

This expression has no actual quantitative meaning, it is merely a symbolism to emphasize the axiom that the effect a given process has on the activity for which it is intended depends not only on the goodness of the process itself, but also on how widely it is used or accepted. In the present context, the first couple of items listed above impact the quality factor; the last two the acceptance factor.

The ability to meet the above requirements truly differentiate between a research CFD method and one that is an engineering and design tool: for design applications, a CFD method must meet all above requirements with a reasonable degree of completeness. It is not uncommon to confuse research feasibility with design application readiness. It is the professional responsibility of the CFD code developer to fully acknowledge this difference and make it clear to the non-expert.

Unfortunately, there is a tendency amongst CFD researchers to greatly underestimate, or completely disregard, the important effort of turning a research, or pilot, code into a practical engineering tool.

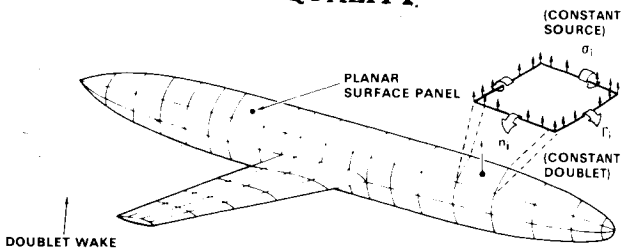
Thus, the principal challenge posed by design applications to CFD practitioners is to convert what is feasible in principle (i.e., feasible in a research laboratory environment) into practical engineering tools that meet all four requirements listed at the beginning of this section.

Some of the principal CFD codes for aircraft design application in use at Lockheed are described in the following sections. These codes have been singled out for discussion herein on the grounds that they have been or are being developed specifically to be practical engineering tools. As in most engineering disciplines, there is a hierarchy in CFD methodology, namely, the various computational codes can be classified according to levels of increasing complexity and accuracy. This also implies different requirements in terms of computational resources. The codes to be discussed, all of them in practical use at Lockheed, span the CFD methodological hierarchy.

LINEAR CFD METHODOLOGY: PANEL METHODS

Methods based on linear inviscid flow assumptions are more fully developed than nonlinear methods, and they are less demanding of computer resources. The most common linear methods are the so called panel methods. The majority of these methods are limited in the flow physics they can model: they cannot deal with shock waves, transonic flows, and separated flows. On the other hand, they are capable of modeling all geometric features of the vehicle, including the most intricate details. At Lockheed, the advanced low-order panel method QUADPAN for subsonic and supersonic flows is the principal tool for linearized analysis, Figure 1. It has been under development and in use for a number of years, Reference 2.

ORIGINAL PAGE IS
OF POOR QUALITY



- WORK WITH PERTURBATION POTENTIAL, ϕ_p : $\vec{V} = \vec{V}_\infty + \nabla\phi_p$
- SOURCE DISTRIBUTION SET BY GREEN'S THEOREM AND CONSERVATION OF LINEARIZED MAS FLUX
- SOLVE FOR DOUBLET DISTRIBUTION
- BOUNDARY CONDITION APPLIED TO FICTITIOUS INTERNAL FLOW
- VELOCITY—2ND ORDER BERNOULLI'S—PRESSURE

Figure 1. Advanced low-order quadrilateral element panel method QUADPAN.

In a panel method, the airplane to be modeled is represented by a large number of small contiguous surface panels. Usually these panels are flat and of a quadrilateral shape. Elementary solutions of the linearized flow equation, known as sources and doublets, are assigned to each panel, Figure 2. The individual strengths of these elementary solutions are determined by the application of the appropriate boundary conditions, e.g., no flow across solid surfaces. The shape of the distribution across a panel of these elementary solutions determine whether the method is low order (e.g., constant or linear variation) or high order (e.g., quadratic variation).

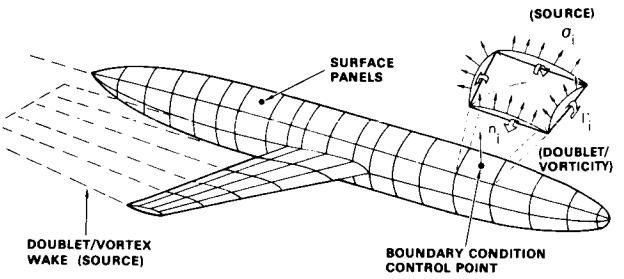


Figure 2. Concept of panel methods

Until recently it was a common belief that a high order distribution was required for better accuracy. The use of high order distribution imposes a sizeable computational cost penalty. Special numerical techniques, discussed in reference 2, can greatly improve the accuracy and robustness of low order singularity distributions. QUADPAN's advanced low order approach embodies these techniques which give it an accuracy similar to that of the high-order methods but at a fraction of the computational cost. This has been verified in evaluations conducted by NASA and U.S. Air Force researchers, References 3 and 4.

Figure 3 illustrates the result of a typical QUADPAN computation which makes evident the code's ability to model complex geometry. It shows a generic advanced fighter configuration with the corresponding elemental quadrilateral panels used to model the flow about it. The computed pressure distribution, at an angle of attack of 10 degrees at a mach number of 0.90, is shown in a color-coded display. Correlation of predicted force and moment coefficients with wind tunnel data are presented in Figure 4.

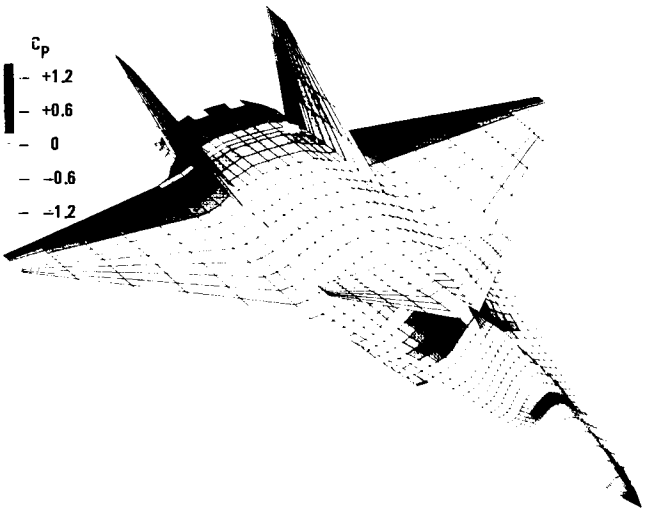


Figure 3. Surface pressure distribution computed by QUADPAN on generic fighter configuration.

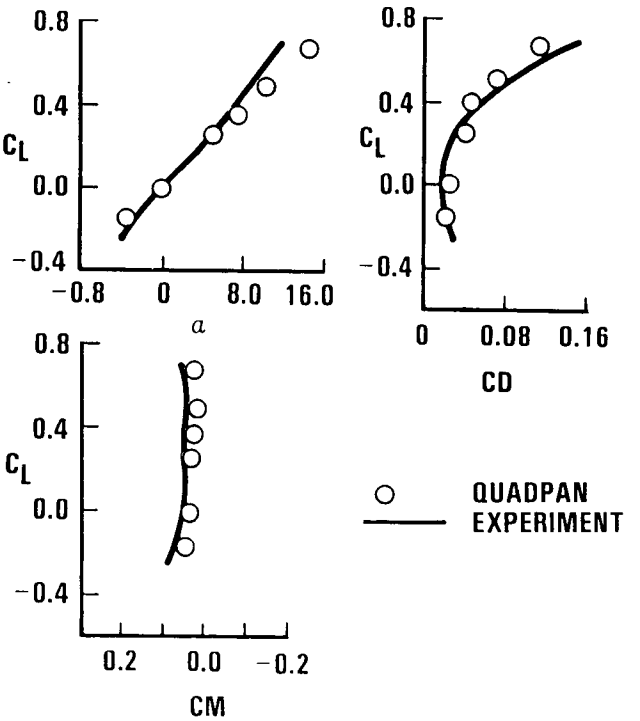


Figure 4. Comparison of QUADPAN predictions with wind tunnel data for generic fighter model.

In spite of the inherent limitations of linearized theory, QUADPAN has proven to be a valuable design tool, Reference 5. This is due to its great geometric capability and to its ease of use and computational efficiency. These factors continue to make linearized methodology the workhorse for conceptual design application. Although the routine use of a linear method such as QUADPAN does not require supercomputer capability, the effectiveness of the method is greatly enhanced by it due to resulting reductions in computational cost and cycle time.

NONLINEAR CFD METHODOLOGY: EULER CODES

Nonlinear methods have been undergoing intensive development. They either solve the full potential or the transonic small perturbation equations for inviscid irrotational flows, or the Euler equations for inviscid rotational flows, or the Reynolds-averaged Navier-Stokes equations for viscous laminar and turbulent flows. Their increased accuracy and realism is attained at the cost of substantially larger computational requirements. Furthermore, the entire space surrounding the vehicle has to be covered with a computational mesh or grid in order to solve the equations of conservation of mass, momentum, and energy throughout the flow field as is required by the nonlinearity of the equations. The generation of this computational grid for arbitrarily complex configurations is a difficult task, and it is of crucial importance for the practical application of these methods.

For flow solution algorithms, the development of time-stepping, finite volume techniques is being emphasized at Lockheed-California Company. These techniques are essentially the same whether the Euler or the Navier-Stokes equations are being solved. The principal difference is the appearance, when solving the Navier-Stokes equations, of shear stress flux terms generated by the viscosity of the fluid and the turbulence of the flow. These flux terms are zero for the Euler equations, which model inviscid flows.

Euler methods are now under advanced development and are beginning to see wide application in the design environment. Navier-Stokes methods are in an earlier stage of development and have only seen pioneering applications. The effective application of these methods in a practical design environment makes the use of supercomputers mandatory.

Although a number of different codes at Lockheed solve the Euler equations, the most advanced in terms of geometric capabilities is the TEAM (for Three-dimensional Euler Aerodynamic Method) code, Figure 5, the development of which is being partially funded by the U.S. Air Force Flight Dynamics Laboratory, References 6, 7, and 8. The TEAM code is a modular computational system consisting of preprocessors, grid generation routines, flow solver, and postprocessors. Its flow solver is based on a variant of the finite-volume time-stepping algorithm proposed by Antony Jameson, et al, Reference 9.

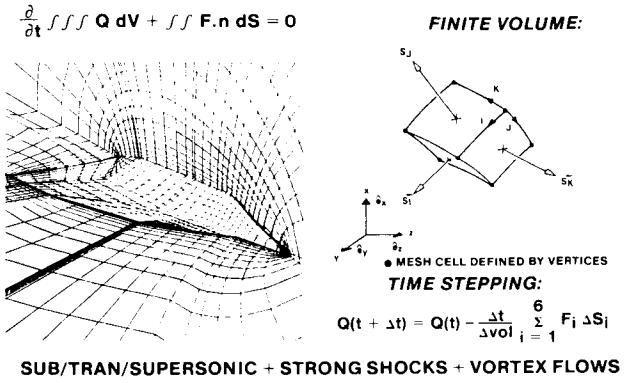


Figure 5. Three-dimensional Euler Aerodynamic Method (TEAM) code.

The ability of the TEAM code to handle arbitrarily complex geometries is due to its zonal structure. This means that the computational domain can be divided into several zones with interfacing boundary surfaces as shown in Figure 6. The various zones can have different grid densities and the interfacing boundary surfaces can accommodate different types of boundary conditions (fluid-to-fluid interface, solid surface, transpiration, free stream). All types of grid topologies (C-H, C-O, O-H, O-O, H-H, and combinations thereof) can be dealt with, Figure 7. Complete airplane configurations including wing, fuselage, canards, horizontal and vertical tails, flow-through nacelles and inlets, etc. are within the realm of the modeling capability of the TEAM code.

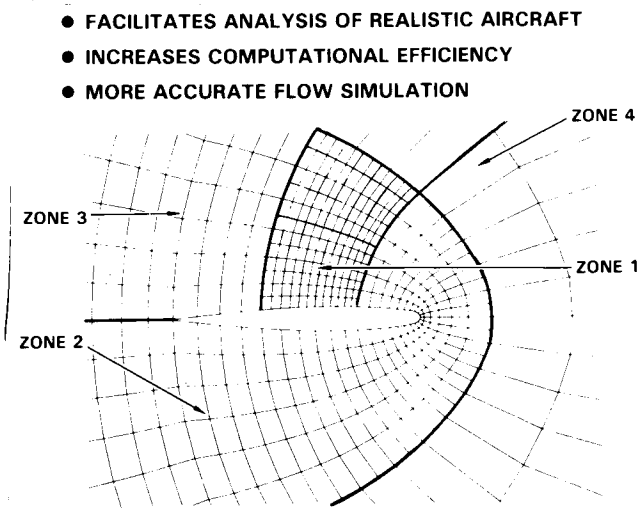


Figure 6. Interfacing zonal grid structure.

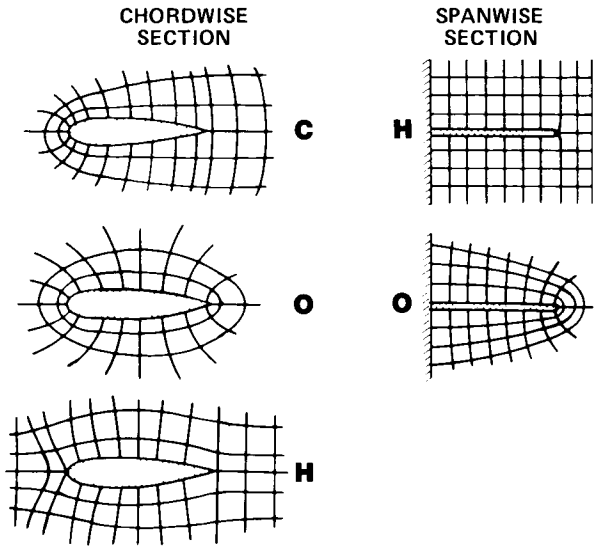


Figure 7. Typical computational mesh topologies.

The code is able to handle subsonic, transonic, and supersonic flows. It can be applied to both steady and unsteady flows, and it automatically captures shock waves and vortex flows induced by sharp edges. The ability to capture vortex flows is of great practical significance, particularly in the design of combat aircraft. Leading edge vortices, like those appearing above a delta wing at high angle of attack, induce strong nonlinear effects which cannot be predicted by classical potential flow theory. Euler methods now offer the capability of predicting these nonlinear effects for complex configurations.

Figure 8 shows the results of a TEAM code computation of the flow about a delta wing at high angles of attack. These results are compared with wind tunnel data and predictions made by classical linear theory. The improvement in accuracy provided by the Euler computations over linear theory is quite evident. Furthermore, the computed velocity field clearly displays the vortex-like structure which characterizes the flow about swept leading edges at high angles of attack.

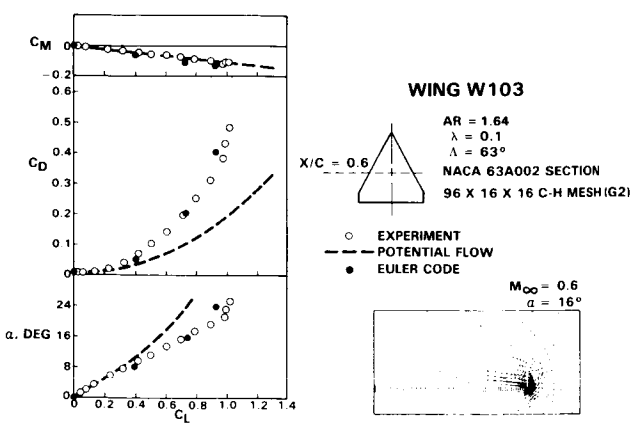


Figure 8. Comparison of computations about delta wing with experimental data.

The solution of the Euler equations appear to do an adequate job of predicting the development of vortex flows induced by sharp or highly swept edges, at least up to the point of vortex burst or breakdown. The geometrical flexibility provided by the TEAM code allows the analysis of complex vortex flow interactions about complete aircraft configurations. Our experience with the TEAM code indicates that in order to obtain reasonably accurate results, fairly dense computational grids are needed. Typically, as many as 400,000 cells may be required for a configuration consisting of a wing, fuselage, and horizontal and vertical tails in a symmetric flight condition.

Some Euler computations show the appearance of unsteadiness in vortical flows at conditions which coincide with the experimental observation of the onset of vortex breakdown. Other qualitative characteristics indicative of the effects of vortex burst have also been observed in these computations. An example of this is illustrated in Figure 9, which shows that the predicted effect of a leading edge strake upon vortex flow development correlates reasonably well with the wind tunnel data. But more study and correlations are needed before ascertaining the usefulness of Euler solutions for the prediction of vortex burst.

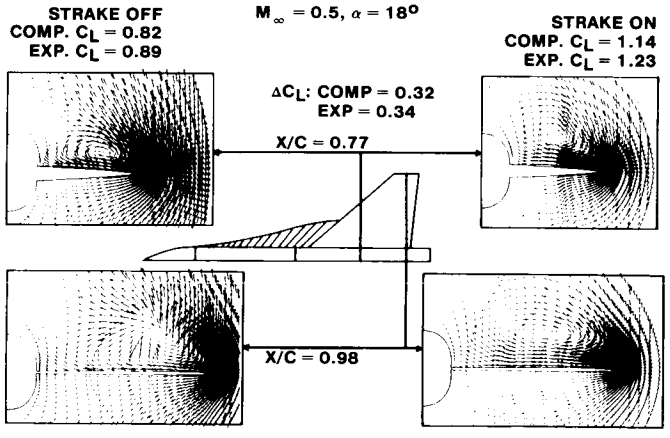


Figure 9. Effect of strake on leading edge vortex flow.

The TEAM code is also being successfully used in high supersonic mach number applications. Figure 10 presents the surface pressure distribution on one of the waverider configurations of Reference 10 at a mach number of 6 and at 4 degree angle of attack. Correlations of the computed pressure distributions with the available wind tunnel data are shown in Figure 11.

The value of Euler computations in design applications have been amply demonstrated in many cases. To what extent they will be able to satisfy the most press-

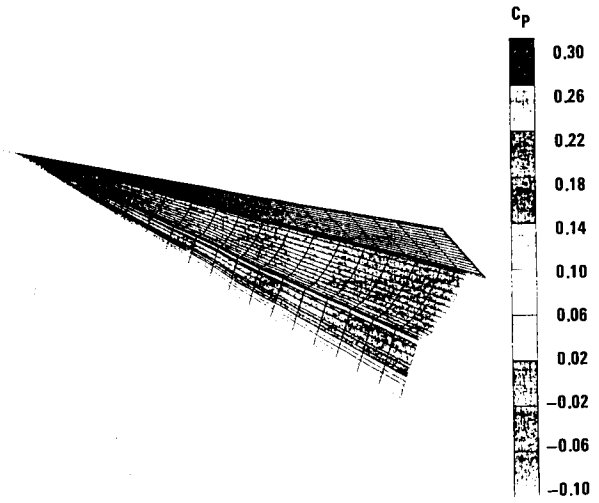


Figure 10. Surface pressure distribution computed by TEAM code on waverider configuration at Mach number = 6 and 4-degree angle of attack.

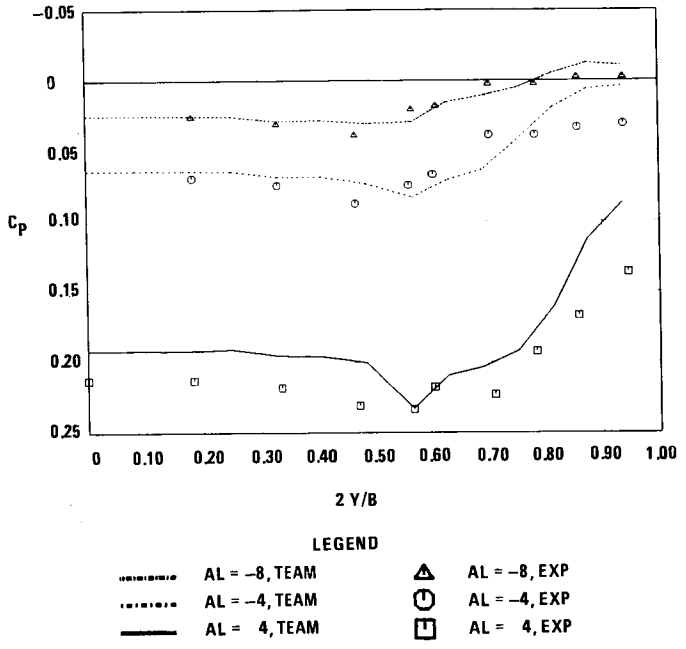


Figure 11. Comparison of predicted lower surface pressure distribution with wind tunnel data.

ing prediction requirements posed by the design of advanced aircraft is far from being fully established. Much work, particularly in terms of advanced computations and correlations with appropriate high quality experimental data, has to be done before the realm of practical applicability and validity of Euler solutions can be determined.

NONLINEAR CFD METHODOLOGY: NAVIER-STOKES CODES

The solution of the Navier-Stokes equations offer the potential of modeling all the physics of a viscous compressible fluid. The calculation of the viscous effects from first principles for all levels of turbulence is presently beyond the realm of practical feasibility due to the lack of adequate computing power. Therefore, the Reynolds-averaged form of the equations is used in engineering applications. The Reynolds-averaged Navier-Stokes equations include the viscous terms but semi-empirical models must be introduced to represent the flow turbulence.

A wide variety of Reynolds - averaged Navier-Stokes codes are in use at Lockheed, including codes with real gas effects and chemistry. But the most advanced from the point of view of geometric capability is the TRANSAM (for Three-dimensional Reynolds-Averaged Navier-Stokes Aerodynamic Method) code, Figure 12. Basically, it constitutes an extension of the modularized TEAM computational system to which momentum fluxes due to both viscous and Reynolds stresses have been added. Therefore, it possesses geometric flexibility and generality similar to those of the TEAM code, and it is equally able to analyze subsonic, transonic, and supersonic flows.

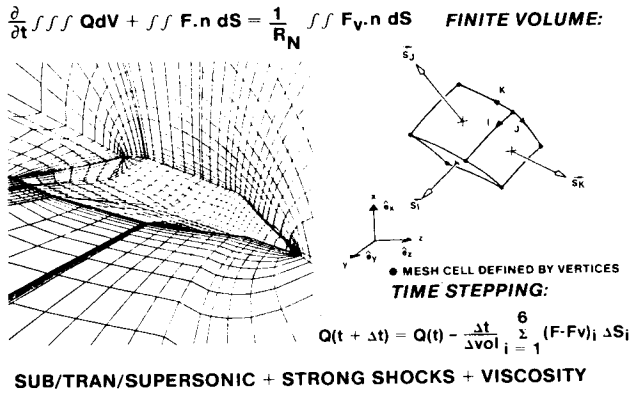


Figure 12. Three-dimensional Reynolds-averaged Navier-Stokes Aerodynamic Method (TRANSAM) code.

In the present version of TRANSAM, either the full Reynolds-averaged or the thin shear layer approximations to the Navier-Stokes equations can be solved at the user's option. The presently implemented turbulence model is an algebraic one, a modified version of the Baldwin-Lomax eddy viscosity model. Since it is architectured in a multiple zonal structure, it is possible to solve different equation types in each zone. For instance, the thin shear-layer approximation to the Reynolds-averaged Navier-Stokes equations can be solved in zones close to solid boundaries where boundary-layer behavior is to be expected; all the shear stress

terms can be accounted for in zones where fully separated flow is likely to occur; and, finally, the Euler equations can be used to model the flow for the remaining zones. This approach yields obvious economies in both computer execution time and memory requirements.

An example of the potential improvement in accuracy that can be obtained with a Navier-Stokes computation is given by the analysis of the supercritical RAE 2822 airfoil shown in Figure 13. Both viscous and inviscid calculations done with TRANSAM are compared with experimental data. The improvement that the viscous computation yields in the prediction of shock location and overall character of the pressure distribution is quite evident. The computed velocity profiles in the boundary layer, shown before and after the shock in Figure 14, show the correct behavior.

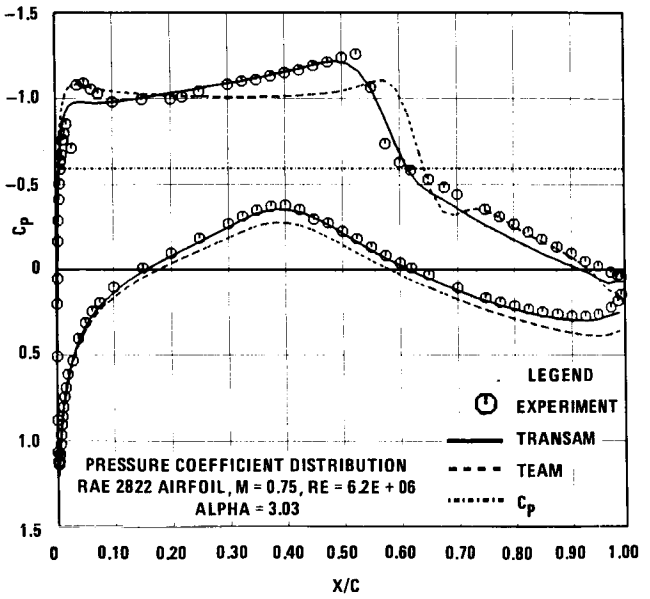


Figure 13. Comparison of computed surface pressure distributions with wind tunnel data for RAE 2822 supercritical airfoil.

We have not yet attempted the computation of separated flows about aircraft at high angles of attack because of lack of adequate memory capacity. Even using the zonal approach provided by TRANSAM, a minimum of 120 megawords of memory would be needed for such a computation. Experience with separated flows about simpler configurations indicates that the presently available algebraic turbulence models are inadequate to predict strongly separated flows with reasonable accuracy and consistency. Whether more sophisticated models (such as the two-equation or Reynolds stress models) will significantly improve the accuracy of separated flow computations remains to be determined.

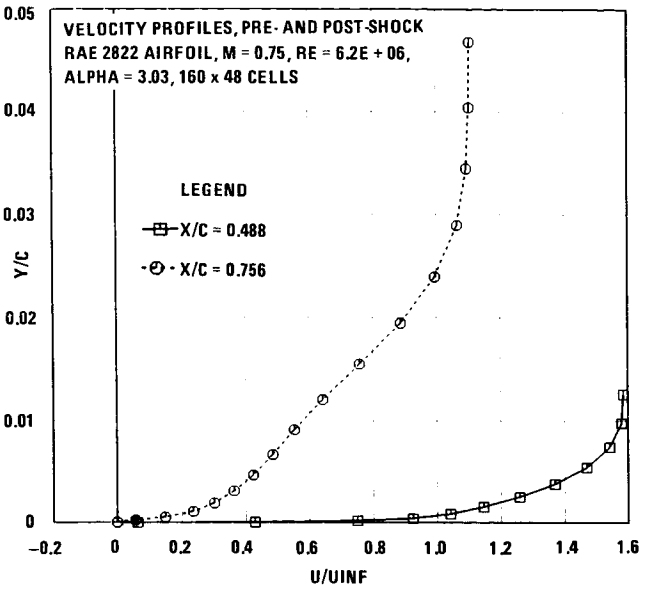


Figure 14. Before and after shock boundary layer profiles computed by TRANSAM code.

In general, the various Navier-Stokes computations of three-dimensional configurations that have been performed to date suffer from lack of adequate computational mesh resolution, and from the limitations of the turbulence models being used. Although some calculations that have recently been presented elsewhere look impressive and appear qualitatively correct, their practical value as predictive and quantitative tools is, at the present time, largely open to question.

In summary, the difficulties hampering the practical application of Navier-Stokes methods, including TRANSAM, are threefold:

1) The lack of general accuracy and reliability of present turbulence models.

2) The difficulty of generating the computational mesh or grid which engulfs the aircraft.

3) The lack of adequate computer resources.

FUTURE DEVELOPMENTS AND CONCLUDING REMARKS

In its present state of capability, CFD provides large benefits in the design process, particularly when used judiciously and synergistically with the wind tunnel. But the full realization of the great potential offered by the application of CFD to aircraft design poses formidable, but not insurmountable, technical challenges. Undoubtedly, the NAS will be highly instrumental in advancing CFD technology the level needed to success-

fully meet them. Some of the major difficulties to be overcome, and probable future developments and trends are highlighted in the remainder of this paper.

The most important obstacle to the ready application of advanced CFD methods to airplane design is the difficulty of generating adequate computational grids for complex three-dimensional configurations in a timely manner. Fortunately, significant progress is being made in this area. Advanced graphics software and hardware developments, e.g., color graphics workstations, are beginning to aid and speed up the grid generation process. Finite-volume zonal methods, such as the ones discussed in this paper, facilitate the grid generation task. In addition, alternate approaches to the treatment of complex geometries are being actively pursued, examples of which are the work discussed in References 11 and 12.

Assuming continuing development, it is very likely that soon it will be possible to generate adequate computational grids about complete and arbitrarily complex aircraft configurations in a matter of hours or days rather than weeks ,or even months, which is the time scale of present grid generation capability.

Turbulence modeling will continue to be the Achilles' heel of Reynolds-averaged Navier-Stokes codes. Reference 13 provides a concise but comprehensive survey of the state-of-the-art in turbulence modeling. This survey makes obvious that many difficulties remain. But in spite of the present short-comings of turbulence models, Navier-Stokes codes will play an increasingly useful role in many applications. Furthermore, turbulence modeling is one area where the NAS offers great potential for advancing the state-of-the-art.

The availability of a wide variety of more capable scientific computers and supercomputers, coupled with advances in numerical solution algorithms, will greatly accelerate the application of Euler and Navier-Stokes methods to the solution of airplane design problems. Computers ranging in speed from 100 to 10,000 million instructions per second and in memory capacity from 8 to 256 million words will become commercially available at competitive prices in the near future.

The analysis and optimization of multidisciplinary interactions will become commonplace thanks to the capabilities of CFD and the foreseen increases in computer power and cost-effectiveness. Coupling CFD with the computational methodologies of other disciplines such as structures, propulsion, dynamics, and flight controls will allow the engineer to optimize not only the aerodynamics but also the structural components, propulsion and control systems, etc., in a fully interactive and integrated manner. Great performance and cost benefits are likely to result from the synergistic effects of this interactive multidisciplinary optimization.

Finally, supercomputers and wind tunnels are complementary and not exclusionary tools. This has been repeated many times, but the more experience we gain with CFD applications, the more evident it becomes. It is also true that to further advance CFD, more high-quality experimental data are required.

REFERENCES

1. Miranda, L. R. "Application of Computational Aerodynamics to Airplane Design," AIAA Journal of Aircraft, Vol. 21, No. 6, pp 355-370, June 1984.

2. Youngren, H. H.; Bouchard, E. E.; Coopersmith, R. M. and Miranda, L. R. "Comparison of Panel Method Formulations and its Influence on the Development of QUADPAN, an Advanced Low Order Method," AIAA Paper 83-1827, AIAA Applied Aerodynamics Conference, Danvers, Massachusetts, July 13-15, 1983.

3. Margason, Richard J.; Kjelgaard, Scott O.; Sellers III, William L.; Morris, Jr., Charles E. K., "Subsonic Panel Methods - A Comparison of Several Production Codes," AIAA Paper 85-0280, AIAA 23rd Aerospace Sciences Meeting, Reno, Nevada, January 14-17, 1985.

4. Strang, W. Z.; Berdahl, C. H.; Nutley, E. L. and Murn, A. J., "Evaluation of Four Panel Aerodynamic Prediction Methods (MCAERO, Pan Air, QUADPAN, and VSAERO," AIAA Paper 85-4092, AIAA 3rd Applied Aerodynamics Conference, Colorado Springs, Colorado, October 14-16, 1985.

5. Johnston, C. E.; Youngren, H. H., and Sikora, J. S., "Engineering Applications of an Advanced Low-Order Panel Method," SAE Paper 851793, SAE Technical Paper Series, Aerospace Technology Conference & Exposition, Long Beach, California, October 14-17, 1985.

6. Raj, P. R. and Long, L. N., "An Euler Aerodynamic Method for Leading-Edge Vortex Flow Simulation," NASA Conference Publication 2416, pp. 263-282, Hampton, Virginia, October 8-10, 1985.

7. Raj, P. R.; Sikora, J. S. and Keen, J. M., "Free-Vortex Flow Simulation Using a Three-Dimensional Euler Aerodynamic Method," ICAS Paper 86-1.5.2, 15th Congress of the International Council of the Aeronautical Sciences, London, United Kingdom, September 7-12, 1986.

8. Raj, P. R. and Brennan, J. E., "Improvements to an Euler Aerodynamic Method for Transonic Flow Analysis," AIAA Paper 87-0040, AIAA 25th Aerospace Sciences Meeting, Reno, Nevada, January 12-15, 1987.

9. Jameson, A.; Schmidt, W. and Turkel, E., "Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes," AIAA Paper 81-1259, 14th Fluic and Plasma Dynamics Conference, Palo Alto, California, June 23-25, 1981.

10. Jones, Kenneth M., "Application of a Supersonic Full Potential Method for Analysis of Waverider Configurations," NASA Technical Paper 2608, Langley Research Center, Hampton, Virginia, 1986.

11. Erickson, L. L. and Strande, S. M., "PAN AIR: Evolution and New Directions . . . Transonics for Arbitrary Configurations," AIAA Paper 83-1831, AIAA Applied Aerodynamics Conference, Danvers, Massachusetts, July 13-15, 1983.

12. Jameson, A., Baker, T. J., and Weatherill, N. P., "Calculation of Inviscid Transonic Flow over a Complete Aircraft," AIAA Paper 86-0103, AIAA 24th Sciences Meeting, Reno, Nevada, January 6-9, 1986.

13. Lakshminarayana, B., "Turbulence Modeling for Complex Shear Flows," AIAA Journal, Vol. 24, No. 12, December 1986, pp. 1900-1917.