N89-20932

TRANSONICS AND FIGHTER AIRCRAFT:

CHALLENGES AND OPPORTUNITIES FOR CFD

Luis R. Miranda
Lockheed Aeronautical Systems Company
Burbank, California

SUMMARY

The application of computational fluid dynamics (CFD) to fighter aircraft design and development is discussed. Methodology requirements for the aerodynamic design of fighter aircraft are briefly reviewed. The state-of-the-art of computational methods for transonic flows in the light of these requirements is assessed and the techniques found most adequate for the subject application are identified. Highlights from some "proof-of-feasibility" Euler and Navier-Stokes computations about a complete fighter aircraft configuration are presented. Finally, critical issues and opportunities for design application of CFD are discussed.

INTRODUCTION

Progress in computational transonics, i.e. transonic CFD, has been most impressive in recent years. A measure of this can be obtained by comparing the papers being presented at this symposium with those given at the previous transonic symposium organized by the National Aeronautics and Space Administration (NASA) at the Ames Research Center in February 1981 (ref. 1). In that occasion, papers discussing CFD applications dealt with solutions of either the transonic small perturbation equation or the full potential equation about wing-alone or simple wing-body configurations. Presently, computations of solutions of the Euler equations about fairly complete aircraft configurations are becoming common. Furthermore, several pioneering computations of the Reynolds-averaged Navier-Stokes equations about wing-fuselage, and in some instances more complex configurations, are in progress, and some promising results have already been obtained. Yet, in spite of this rather sensational progress, the application of CFD to the design of fighter aircraft still poses a formidable challenge.

In this paper I intend to explain the magnitude of this challenge and share some of the experience and lessons learned in applying CFD at the Lockheed Aeronautical Systems Company (LASC). First, I will review the major requirements for the aerodynamic design of fighter aircraft. Then, I will briefly describe the computational methods and techniques that we have found most adequate for transonic applications. I will show highlights from Euler and Navier-Stokes flow computations about a complex fighter configuration to illustrate what is presently feasible with the state-of-the-art tools of CFD, but I will also discuss the principal problems and difficulties that the CFD practitioner faces today. This will help underscore the major developments that are needed to make CFD realize its full potential as an effective design tool for fighter aircraft. Finally, I will point out some opportunities for CFD applications that may greatly assist the designer even within the limits of present CFD shortcomings.

FIGHTER AIRCRAFT DESIGN REQUIREMENTS

In general, advanced tactical aircraft must be designed to satisfy an extensive set of performance requirements. Figure 1 illustrates a typical design mission and the corresponding performance requirements for an advanced fighter. Usually, not one but several of these requirements become design drivers, as has already been discussed by Bradley in reference 2. For example, the following combination of performance requirements may dictate the design solution:

- Transonic cruise
- Sustained transonic maneuver
- Supersonic cruise
- Transonic acceleration

This multiple design point requirement is in contrast with transport aircraft, where usually a single performance consideration, i.e., cruise efficiency, becomes the paramount design driver. These multiple design points involve both attached and separated flow conditions (fig. 2), whereas cruise efficiency implies well behaved, attached flow. Furthermore, future fighter aircraft will be required to operate and, therefore, be controllable at very high angles of attack, beyond the onset of flow separation and maximum lift. Consequently, it will be crucial to determine how configuration details affect the onset of, and the behavior of the flow after separation. Major design features such as forebody shape, wing geometry, layout of control surfaces, etc., may be driven by the impact of these features upon the aerodynamic forces and moments at angles of attack beyond stall rather than by their effects in attached flow conditions.

A representative example of how relatively subtle configuration differences can greatly change the aerodynamic characteristics at high angles of attack is given by the effect of forebody shape on directional stability (fig. 3). The data presented in figure 3 were obtained from a number of fighter configurations with their vertical tails removed. All configurations are directionally unstable at low angles of attack and remain so until approximately 25 degrees. Above 25 degrees, cross-sectional shape effects become evident: the horizontal ellipse cross-sectional shape forebody turns directionally stable whereas the vertical ellipse forebody increases its instability.

In addition to the multiplicity and difficulty of flow conditions that the fighter aerodynamicist has to contend with, he must also deal with geometries that are complex and prone to generate strong interference effects among the configuration components (fig. 4). In many cases, the design depends on the maximization or tailoring of these interference effects as, for instance, in close-coupled canard configurations.

Airframe/propulsion integration is of paramount importance in fighter design, particularly with thrust-to-weight ratios equal to or greater than 1. This calls for flow computations at the inlet face, in inlet-diffuser geometries with duct offset and drastic cross-sectional shape variations, and about complex afterbody geometries.

Last but not least, all the related problems of weapons carriage and release, namely, store loads, separation characteristics, weapons bay cavity flows, etc., have to be addressed.

The designer of fighter aircraft must be able to deal with the nonlinear and difficult flow problems and complicated geometries discussed above because they are predominant in fighter aerodynamics. In the past, his only means of doing this was the wind tunnel. But now, advanced CFD methodology is beginning to offer the possibility of dealing with the vexing problems of nonlinear aerodynamics. Although the jury is still out regarding the ultimate value of CFD as a fighter design tool, rapid progress is being made and valuable lessons are being learned in our efforts to apply CFD to fighter aircraft. The CFD methods and techniques that we have found to be most appropriate for fighter design application are discussed in the following section.

COMPUTATIONAL TRANSONICS FOR FIGHTER AIRCRAFT

The experience at LASC with CFD methods for transonic flow analysis has covered a broad spectrum of the presently available methodology, as shown in table 1. Codes based on either the transonic small perturbation (TSP) equation or the full potential (FP) equation, though they may be suitable for some limited and specific applications, have been found to be inadequate for general fighter aircraft application. The reason for this is obvious: most fighter aerodynamic problems violate the assumptions of small perturbations (the fundamental assumption for the TSP equation) and irrotationality (inherent to both the TSP and FP formulations). Vortex flows play a preponderant role for highly swept leading edges and sharp edges, features which are commonly found on fighter configurations. An example of the difference between potential flow and Euler solutions for this class of problem is illustrated in figure 5. This example clearly underscores the inadequacy of potential flow methods for the analysis of free vortex flows.

The need to deal with vortex flows, strong shock waves, and, eventually, separated flows, has led us to concentrate on Euler and Reynolds-averaged Navier-Stokes (R-A N-S) methods in our transonic CFD work. At the present time, our experience with Euler codes is much more extensive than with R-A N-S codes. But thanks to the recent availability of supercomputers with very large memory capacity, we are rapidly expanding our R-A N-S experience base.

The majority of practical methods for solving the Euler and Navier-Stokes equations are based on either finite difference or finite volume numerical approaches. Finite difference schemes result from the discretization of the partial differential formulation of the equations of fluid flow. Finite volume schemes are derived by discretizing the integral formulation of the flow equations. Although in principle both approaches are equivalent, their actual numerical implementations involve differences which make finite volume methods more robust for obtaining flow solutions about the type of geometries characteristic of fighter aircraft. This is because finite difference methods, due to the differential nature of their formulation, are more sensitive to boundary singularity problems, such as sharp edges and surface slope discontinuities. Accordingly, we have emphasized the development and application of explicit, time-stepping, finite-volume techniques for the solution of both the Euler and Navier-Stokes equations. These techniques are based on the work of Jameson et al (refs. 3-4).

Experience with realistically complex and complete aircraft geometries has taught us that codes requiring single global computational grids are woefully inadequate. For general applications, codes must be able to handle multiple zonal grid blocks (fig. 6). Otherwise, computer memory requirements increase substantially, and more seriously, the grid generation task, which is already difficult and time-consuming, becomes much more complicated, and in some instances, practically impossible.

The two approaches commonly used for generating multiple zonal grid blocks are 1) grid embedding, and 2) grid patching (fig. 7). In the grid embedding approach, computational grids are generated about the principal configuration components (e.g., wing, nacelle, fuselage, etc.) as if they were in isolation, and then they are assembled together in an overlapping fashion. In the grid patching approach, the various zones are separated by interfacing surfaces and there is no grid overlap between the zones. Although grid embedding facilitates the use of boundary conforming orthogonal grids, it has some serious drawbacks: it requires special interpolation schemes for the overlapped regions, it leads to cumbersome data structures, and it makes conservation of the pertinent flow quantities difficult to preserve. Grid patching, on the other hand, makes boundary conforming grids with reasonable orthogonality more difficult to achieve, but instead, it possesses some very good attributes for practical application such as well ordered data structures, good conservation properties, and good convergence and accuracy characteristics. Because of these attributes, we have elected to work with grid patching instead of grid embedding.

The above considerations have guided the development at LASC, and under partial funding from the USAF Flight Dynamics Laboratory, of the TEAM (Three-dimensional Euler Aerodynamic Method) and TRANSAM (Three-dimensional Reynolds-Averaged Navier-Stokes Aerodynamic Method) codes (refs. 5-8). These codes incorporate cell-centered finite volume flow solvers with explicit multistage Runge-Kutta time marching. They can operate on multiple block zonal grids of arbitrary topology with the three different types of zonal interfaces illustrated in figure 8. The treatment of the zonal interfaces has been formulated along the lines proposed by Rai (ref. 9).

The TEAM code solves the Euler equations for inviscid flow. These equations contain all of the continuum flow physics except for viscosity. Rotational flows, such as vortex flows, and strong shock waves can be simulated with the Euler equations. The TRANSAM code is an extension of the modularized TEAM computational system to which momentum fluxes due to both viscous and Reynolds, namely, turbulent flow stresses have been added. 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 user can also choose between algebraic and two-equation turbulence models. Because of its zonal architecture, different equation sets can be solved in different zones. For instance, the thin shear-layer 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, essentially inviscid, zones. This approach yields substantial savings in both computer execution time and memory requirements.

ADVANCED CFD APPLICATION HIGHLIGHTS

With the TEAM and TRANSAM codes it is now feasible to perform both inviscid (TEAM) and viscous (TRANSAM) flow computations about arbitrarily complex and complete aircraft configurations at subsonic, transonic, and supersonic mach numbers. "Proof-of-feasibility" computations have recently been performed at LASC using the supersonic vertical/short take-off and landing (V/STOL) fighter concept of reference 10, which is known as the Advanced Nozzle Concept (ANC) configuration. This configuration was selected for these "proof-of-feasibility" computations due to its challenging geometric complexity (fig. 9). The computations were done on the LASC Cray X-MP/24 supercomputer.

A partial view of the grid used for the inviscid Euler computations is shown in figure 10. A total of 288,750 finite volume cells was required to cover half the configuration, which is symmetrical about the x-z plane. These cells were distributed in 25 different zones; the largest zonal block containing 35,464 cells, and the smallest one 252 cells.

The surface pressure distribution computed by the TEAM code, at an angle of attack of 4.8 degrees and mach number of 1.2, is shown in color-coded displays in figures 11 and 12. The computational mesh on the airplane surface is also visible in these figures. In the TEAM computations the nacelles were treated as flow-through ducts, in other words, power effects were not simulated. The differences between the numerically computed and wind tunnel measured values were 5 percent for lift and 8 percent for pitching moment. Drag correlation has not yet been attempted due to the lack of modeling of the wind tunnel model support system. Reasonable correlation was obtained for the one wing station, just outboard of the nacelles, for which some rather sparse experimental surface pressure distribution data were available.

For the Navier-Stokes computations, 521,224 cells were required to cover half the configuration. These cells were distributed in 27 different zones; the largest zonal block containing 42,312 cells, and the smallest one 480 cells. The total number of cells was constrained by computer memory and processing time considerations. A larger number of cells is desirable for an accurate viscous computation. In particular, because of computer capacity constraints, the grid coverage over most of the fuselage is considered quite inadequate for viscous flow simulation.

The surface pressure distribution computed by the TRANSAM code at the same flow condition shown previously for the Euler case (angle of attack of 4.8 degrees and mach number of 1.2) is presented in color-coded displays in figures 13 and 14, which also illustrate the computational surface grid. Representative boundary layer velocity profiles for the wing upper surface are shown in figure 15. The Reynolds number, based on the wing mean aerodynamic chord, was 6.5 million. The thin shear layer approximation to the Navier-Stokes equations was used for this computation. Fully turbulent flow was assumed, turbulence being modeled with a modified Baldwin-Lomax eddy vicosity. Little difference can be observed between the viscous and inviscid computations for this case. Some minor improvement was observed in the correlation of computed lift and pitching moment with experimental data.

These computations will be continued as soon as a Solid-state Storage Device (SSD) is attached to the LASC Cray supercomputer, adding 128 megawords of fast access memory. This will allow increasing the grid density for adequate simulation of viscous and separated flow characteristics. Similar computations to study the adequacy of TRANSAM for high angle of attack flows about fighter configurations are planned for the NASA Ames Research Center Numerical Aerodynamic Simulator (NAS).

CRITICAL ISSUES FOR DESIGN APPLICATION

The potential capability of CFD methodology, which the preceding examples give a glimpse of, is remarkable. The challenge is to convert this capability from potential into actual within the constraints imposed by the design environment. To do this successfully, much work remains to be accomplished. Our recent experience underscores the critical importance of the following four major issues:

1) Grid sensitivity: Results of Euler and Navier-Stokes computations - even those based on supposedly robust schemes such as finite volume - display a

high degree of sensitivity to the characteristics of the computational grid: density, distribution, and skewness. In many cases this sensitivity is more pronounced than that due to the type of mathematical model being used, e.g., Navier-Stokes versus Euler equations. Grid characteristics also affect the convergence of the solution process for time-stepping methods.

Two examples are used here to underline the impact of grid features on the quality of the solution. The first one (fig. 16) shows the effect of grid density (coarse versus fine) on the surface pressure distribution on the Onera M6 wing near the tip. The second one, internal flow computations for a subsonic diffuser (fig. 17), illustrates the effect of grid point distribution. Three computations were carried out with an inlet Mach number of 0.72: an Euler (TEAM) and two turbulent Navier-Stokes (TRANSAM) computations. The latter were done at a Reynolds number of about 1.7 million (based on duct diameter) using an algebraic eddy viscosity model. The first viscous computation was performed on the same grid used for the inviscid computation. This grid had some clustering of points near the solid boundaries; otherwise, it was uniformly spaced. Grid clustering near the walls was increased twofold for the second viscous computation, but the total number of cells was kept constant (31 in the radial direction). The corresponding velocity profiles (fig. 17) show the strong influence of grid spacing; it is quite obvious that the results from the first viscous computation are physically unrealistic.

To resolve the grid sensitivity problem we must come up with means of a) determining the adequacy of a computational grid for reliable results, and b) adjusting the grid to become adequate in the areas found deficient. Computationally-adaptive grids offer much promise in this respect although the development of robust algorithms for three-dimensional and multiblock zonal application will not be easy.

2) <u>Turbulence modeling</u>: Turbulence modeling is the Achilles' heel of Reynolds-averaged Navier-Stokes codes. Reference 11 provides a concise but comprehensive survey of the state-of-the-art in turbulence modeling. This survey makes obvious that many difficulties remain. Experience with separated flows indicates that the presently available algebraic turbulence models are inadequate to predict strongly separated flows with reasonable accuracy and consistency. Usually, extensive code calibration is required to reproduce experimental results. Furthermore, these calibrations tend to be restricted to relatively narrow classes of problems. The degree by which more sophisticated models (such as the two-equation or Reynolds stress models) will improve the accuracy of separated flow computations remains to be determined.

3) <u>Timeliness</u>: The time from "blueprint" to first satisfactory nonlinear CFD solution is presently totally inadequate for design application. The Euler and Navier-Stokes solutions about the supersonic V/STOL fighter shown above required about six months of effort to obtain. Similar computations conducted on an F-16 configuration (ref. 12) have taken about a year to complete. The principal cause for this is the difficulty of generating adequate computational grids for complex three-dimensional configurations. 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, like the ones discussed in this paper, facilitate the grid generation task. Application of artificial intelligence and expert systems technology will probably help accelerate grid generation.

In addition, alternate approaches to the treatment of complex geometries are being actively pursued, examples of which are the work discussed in references 13 and 14. These alternate methods allow the use of either cartesian non-boundary-conforming or non-structured grids.

For Navier-Stokes codes the timeliness issue is further aggravated by the extensive calibration and numerical experimentation which are required to overcome the shortcomings of present turbulence models.

4) **Validation**: A code can be considered validated when its accuracy and range of validity have been determined sufficiently well to be applied, without calibration, to the problem of interest with a high degree of confidence. Validation of flow field solvers, such as the Euler and Navier-Stokes codes, requires experimental data of special type and quality which presently are very scarce. Consequently, very few, if any, of the advanced CFD codes can be considered to be validated in the true sense of the word. Yet, validation is a most important factor in determining the acceptance of a CFD code by the design community.

OPPORTUNITIES FOR CFD APPLICATION

In the light of the previous considerations, it may be argued that, at the present time, nonlinear CFD methodology is more postdictive than predictive in nature. This may be so due to the difficult problem posed by fighter aircraft design requirements. Lest we forget, wind tunnels also face difficulties that, although different in nature, are equally serious: wall and support interference, Reynolds number effects, difficulty or impossibility of simulating unsteady motions, etc. Furthermore, CFD should be viewed not as a tool to replace the wind tunnel, but rather as a tool to complement the wind tunnel. With this in mind, there are many opportunities for valuable application of CFD to fighter design, even within its present limitations. Some of these opportunities are

- **Design**: It has already been pointed out how the design of a fighter aircraft depends on predicting aerodynamic characteristics for flow conditions for which the present accuracy of CFD is questionable. Yet, in the design or synthesis process, oftentimes it is sufficient to determine qualitatively, rather than quantitatively, which of the design options under consideration is the best. In this sense, CFD methodology is quite capable. The timeliness issue is the only major obstacle to its full suitability as a synthesis tool.

- **Wind tunnel corrections**: Correcting wind tunnel data for wall and model support interference is well within the realm of CFD capability. With due calibration, correcting for Reynolds number effects should also be quite accurate.

- **Experimental data enrichment**: Postdiction can be quite useful when properly used. CFD can provide flow field description and details that are beyond the practical capability of even the most advanced experimental techniques. Thus, after careful calibration, it can be used to enrich and expand the experimental database and to help diagnose unusual problems uncovered by testing.

- **Airloads prediction**: This is one application area where CFD can be used to great advantage. Nonlinear CFD methodology offers significant improvement in accuracy over classical linearized methods, as the example of figure 18 indicates. With a reasonable amount of calibration based on simpler wind tunnel models, it should be possible to obtain loads data for structural design of adequate accuracy and detail at a much earlier stage in the design process than that provided by the expensive pressure loads models currently used.

- Configuration modification evaluation: The ability of CFD to predict incremental values more accurately than absolute levels is well recognized, and it has been demonstrated on many occasions. Therefore, CFD is a most useful tool in assessing the effects of configuration perturbations, particularly after calibrating the method on the baseline geometry.

These application opportunities by themselves make CFD a valuable tool for fighter design and development. But to take full advantage of them, it is imperative that the time and labor required for the computational grid generation process be greatly reduced.

CONCLUDING REMARKS

CFD is becoming an increasingly powerful tool for the aerodynamic design and analysis of aerospace systems. Fighter aircraft, because of the multiple design point requirements involving difficult flow conditions, present a formidable CFD application challenge. Several "proof-of-concept" computations are beginning to demonstrate the ultimate potential of CFD, but much remains to be accomplished before CFD can be accepted as a fighter aircraft design and development tool with a high level of confidence.

Our experience has helped identify the multiblock zonal, finite volume, time marching flow solvers as the presently preferred approach for fighter aircraft design and development application. It has also highlighted four crucial issues that must be successfully resolved to turn CFD into a practical and reliable design and development tool: grid sensitivity, turbulence modeling, timeliness, and validation. The task is not easy, but it is feasible and the benefits are high.

ACKNOWLEDGEMENTS

The author is indebted to the staff of the Computational and Advanced Aerodynamics Department at the Lockheed Aeronautical Systems Company - Burbank for providing technical assistance and consultation. Especially, the contributions of K. K. Mani, who performed the Euler computations on the ANC configuration, C. R. Olling, responsible for the corresponding Navier-Stokes solutions, and P. Raj were crucial to the preparation of this paper. K. Henry assisted in the preparation of the text and technical art material.

REFERENCES

1. Nixon, D. (editor) : Transonic Aerodynamics. Progress In Aeronautics and Astronautics, Volume 81, 1982.

2. Bradley, R. G.: Practical Aerodynamic Problems - Military Aircraft. Progress in Aeronautics and Astronautics, Volume 81, 1982, pp. 149-187.

3. 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 Fluid and Plasma Dynamics Conference, Palo Alto, California, June 23-23, 1981.

4. Jameson, A., and Baker, T. J.: Solution of the Euler Equations for Complex Configurations. Proceedings AIAA 6th Computational Fluid Dynamics Conference, Danvers, Massachusetts, 1983, pp. 293-302.

5. Miranda, L. R.: CFD Applications: The Lockheed Perspective. NASA Conference Publication 2454, pp. 77-85, Moffett Field, California, March 10-12, 1987.

6. Raj, P., 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., 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 Aerospace Sciences, London, United Kingdom, September 7-12, 1986.

8. Raj, P., 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. Rai, M. M.: A Conservative Treatment of Zonal Boundaries for Euler Equation Calculations. AIAA Paper 84-0164, Reno, Nevada, 1984.

10. Zilz, D. E., and Devereaux, P. A.: Propulsion and Airframe Aerodynamic Interactions of Supersonic V/STOL Configurations. NASA Contractor Report 177343, September 1985.

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

12. Flores, J., Reznick, S. G., Holst, T. L., and, Gundy K.: Transonic Navier-Stokes Solutions for a Fighter-like Configuration. AIAA Paper 87-0032, Reno, Nevada, January 12-15, 1987.

13. Samant, S. S., Burkhart, R. H., Bussoletti, J. E., Everson, B. L., Johnson, F. T., Young, D. P., Erickson, L. L., Madson, M., and Woo, A. C.: TRANAIR: A Computer Code for Transonic Analyses of Arbitrary Configurations. AIAA Paper 87-0034, Reno, Nevada, January 12-15, 1987.

14. Jameson, A., and Baker, T.: Improvements to the Aircraft Euler Method. AIAA Paper 87-0452, Reno, Nevada, January 12-15, 1987.

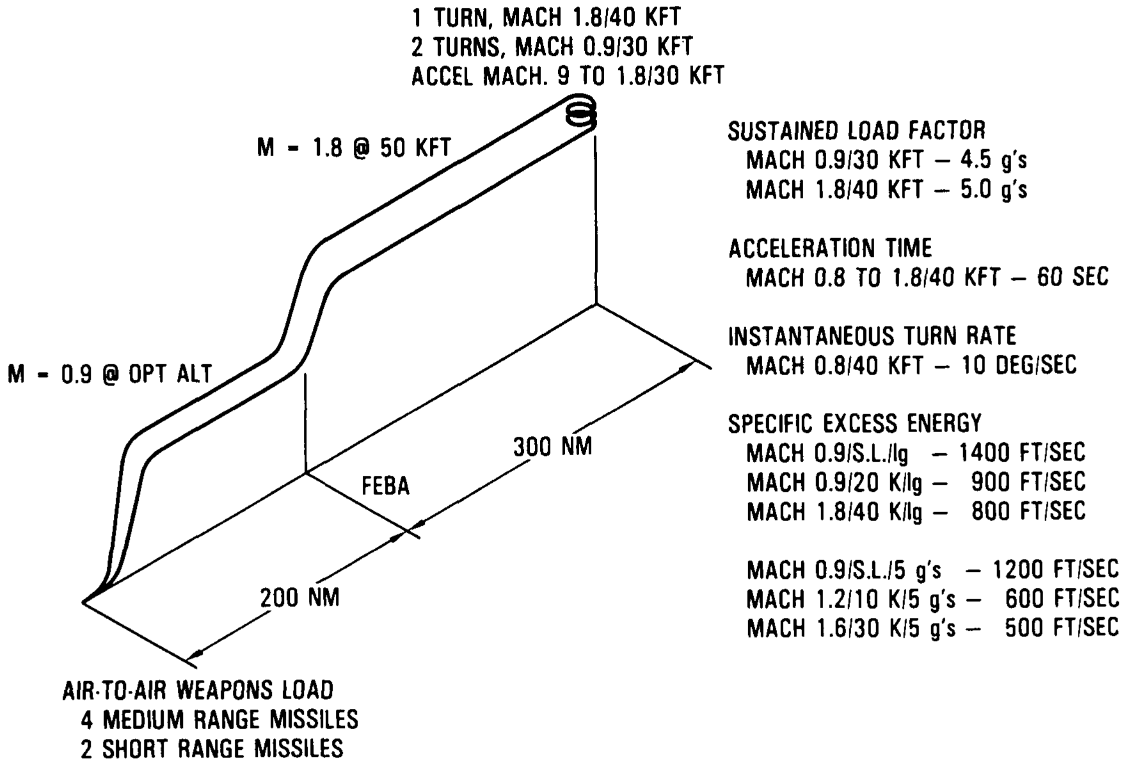

Figure 1 Typical Fighter Design and Performance Requirements.

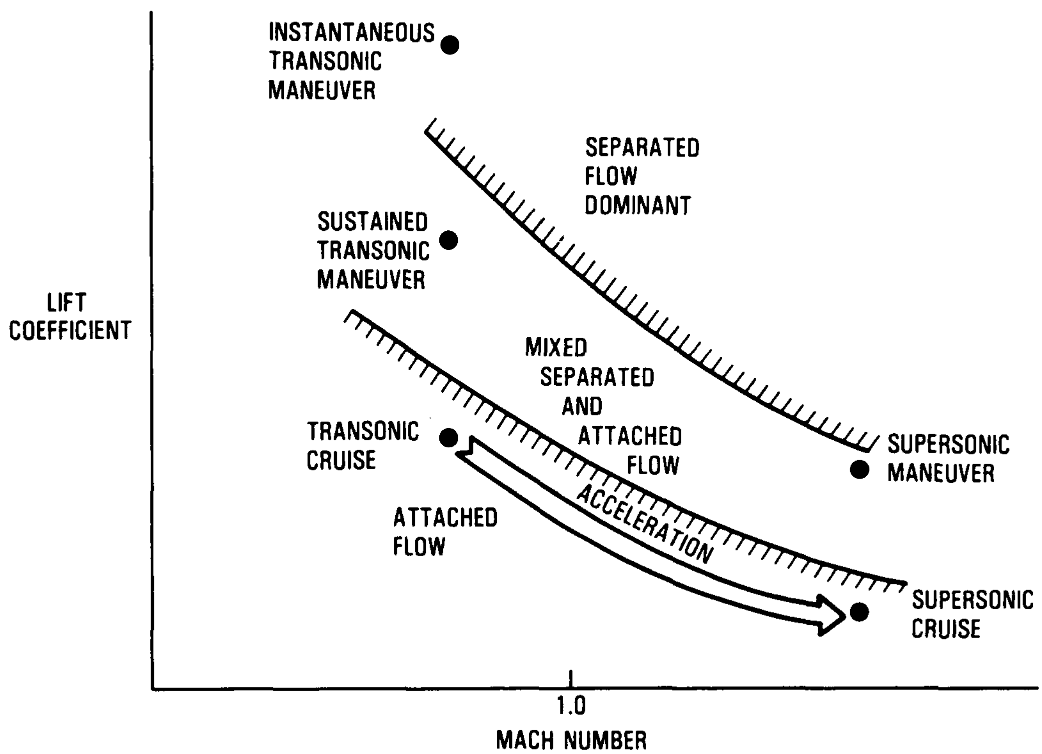

Figure 2 Multiple Design Points and Flow Conditions for Advanced Fighter Aircraft.

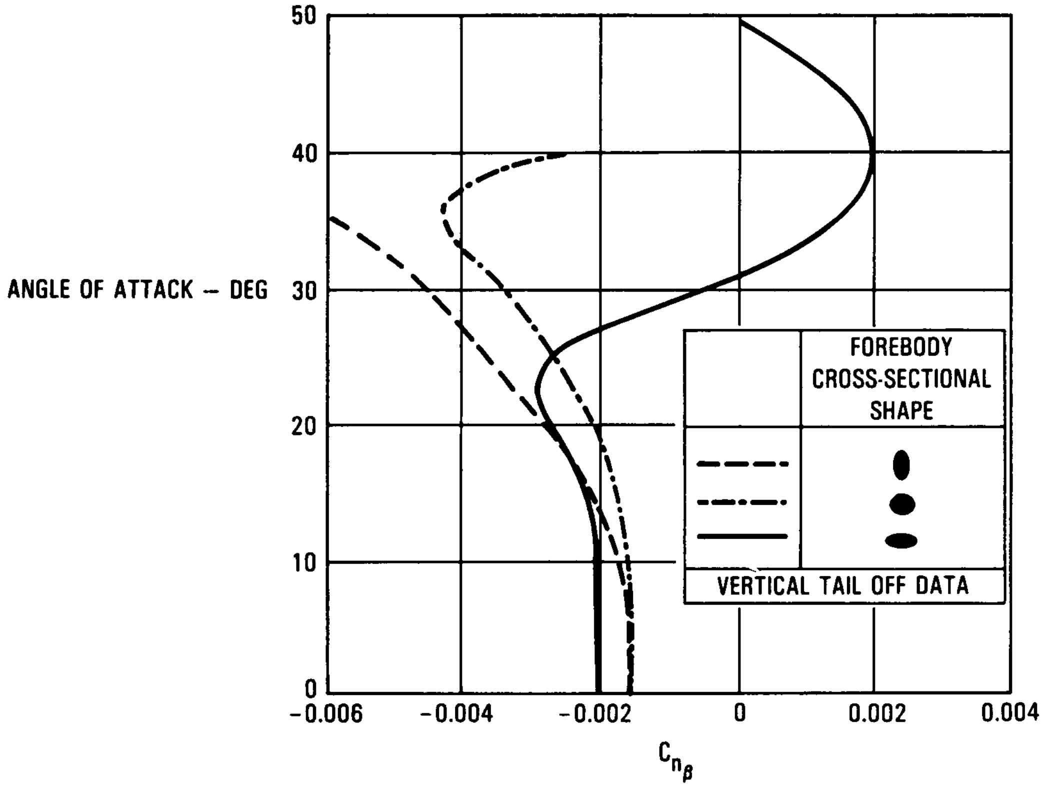

Figure 3 Effect of Forebody Cross-sectional Shape on Directional Stability.

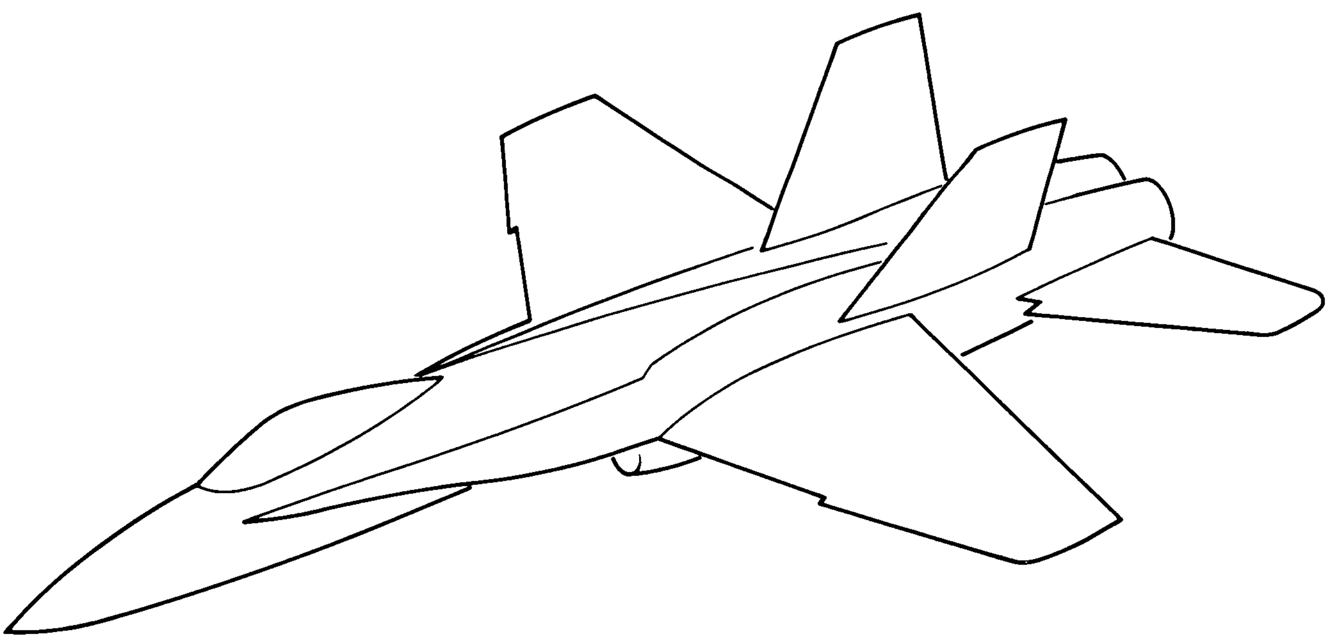

Figure 4 Representative Advanced Fighter Aircraft Configuration.

TABLE 1.- TRANSONIC CFD METHODOLOGY

FLOW EQUATION	CODE USED AT LOCKHEED	SOLUTION APPROACH
TRANSONIC SMALL PERTURBATION	BOPPE (1976)*	Non-conservative finite difference on cartesian, i.e., non-body-conforming grid.
FULL POTENTIAL	FLO-22.5 (1978)*	Non-conservative finite difference on body-conforming grid.
EULER	TEAM (1984)*	Time-marching finite volume. Zonal multiblock body-conforming grid.
REYNOLDS-AVERAGED NAVIER-STOKES	TRANSAM (1986)*	Time-marching finite volume. Zonal multiblock body-conforming grid. Algebraic or 2-eqn. turb. model.

* Year of first application.

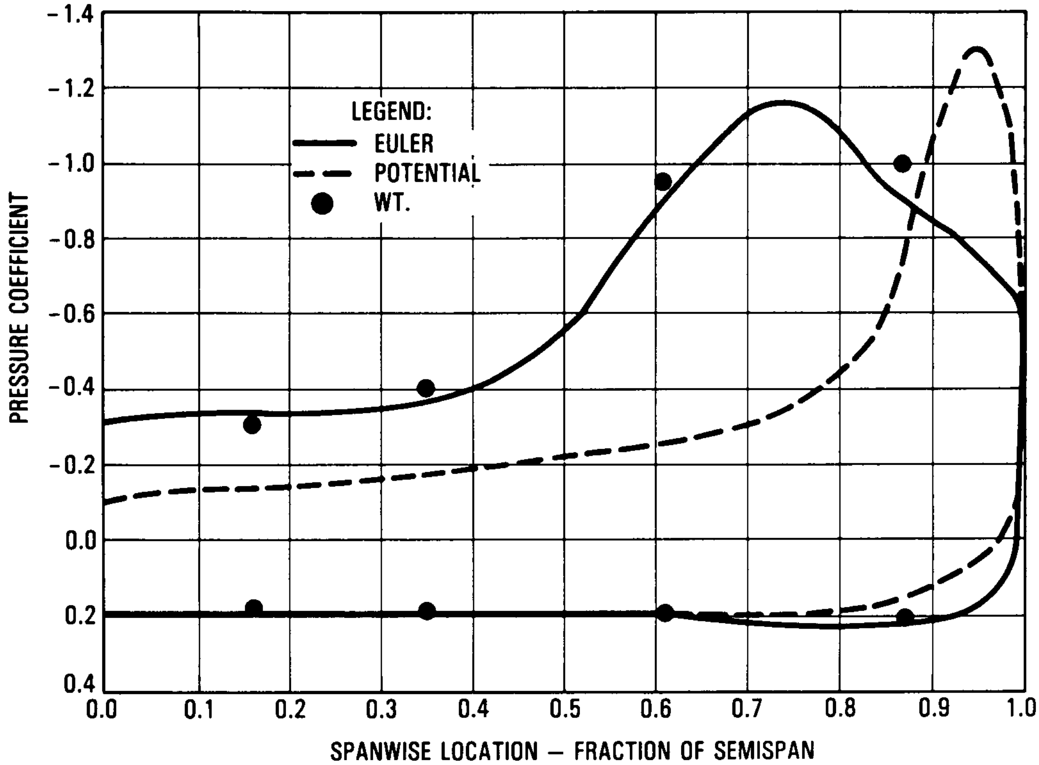

Figure 5 Euler vs. Potential Flow Solution about Arrow Wing-body with Sharp Leading Edge at Mach = 0.85 and Alpha = 15.8 Degrees.

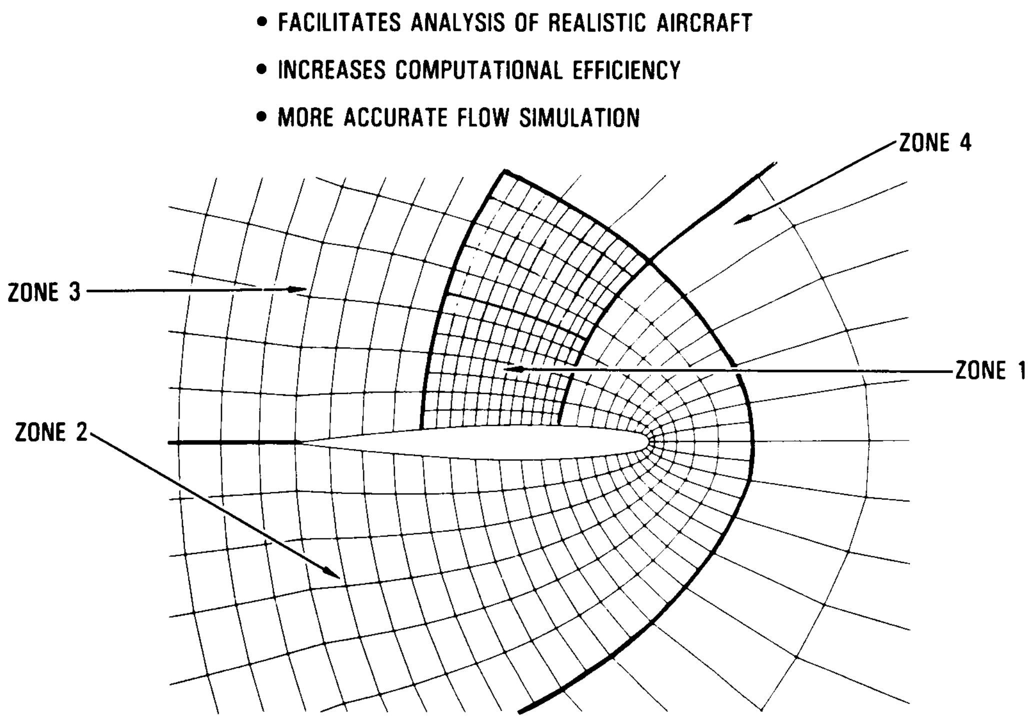

Figure 6 Multiple Zonal Grid Blocks.

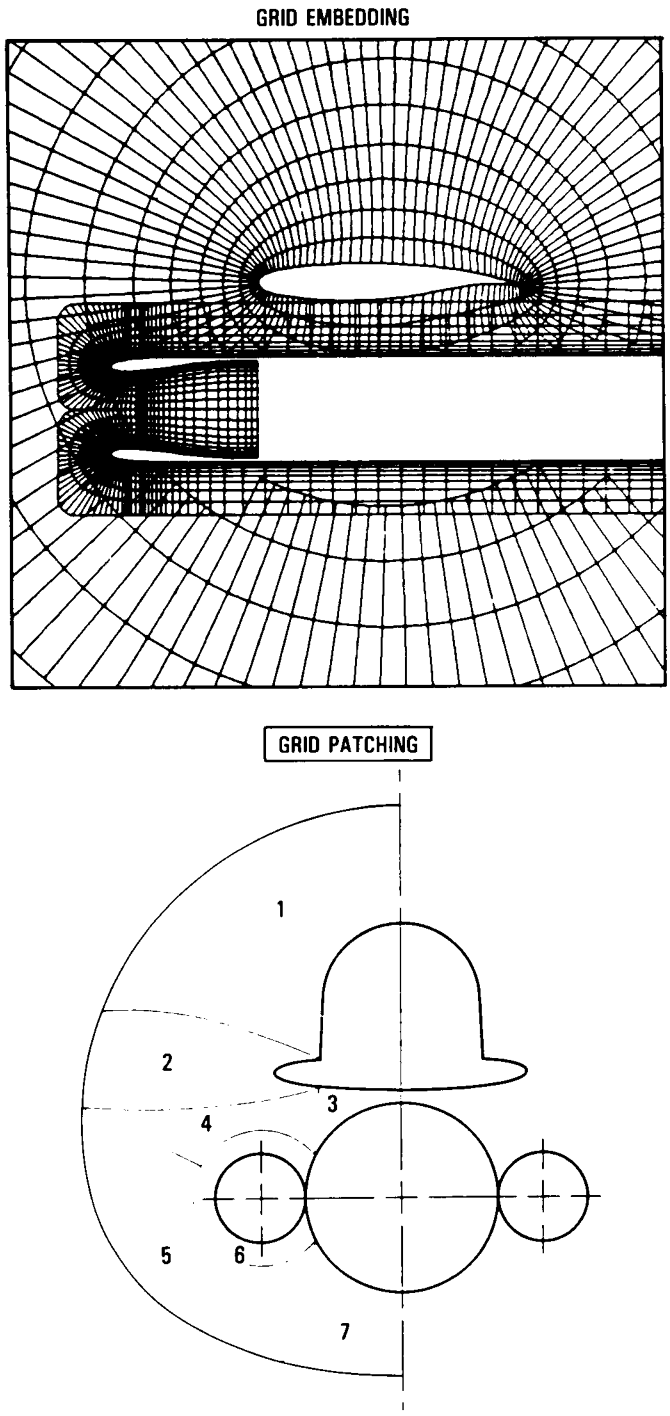

Figure 7 Approaches for Generating Zonal Grids: Grid Embedding versus Grid Patching.

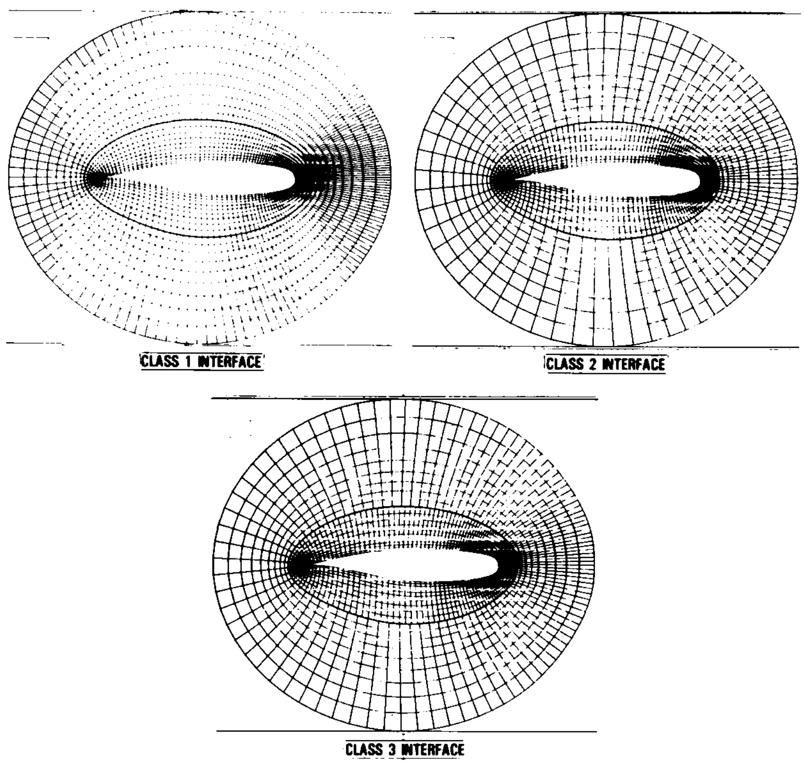

Figure 8 Three Classes of Zonal Interfaces Handled by the TEAM and TRANSAM Codes: 1) One-to-One Correspondence, 2) Integer Correspondence, and 3) Noninteger (Arbitrary) Correspondence.

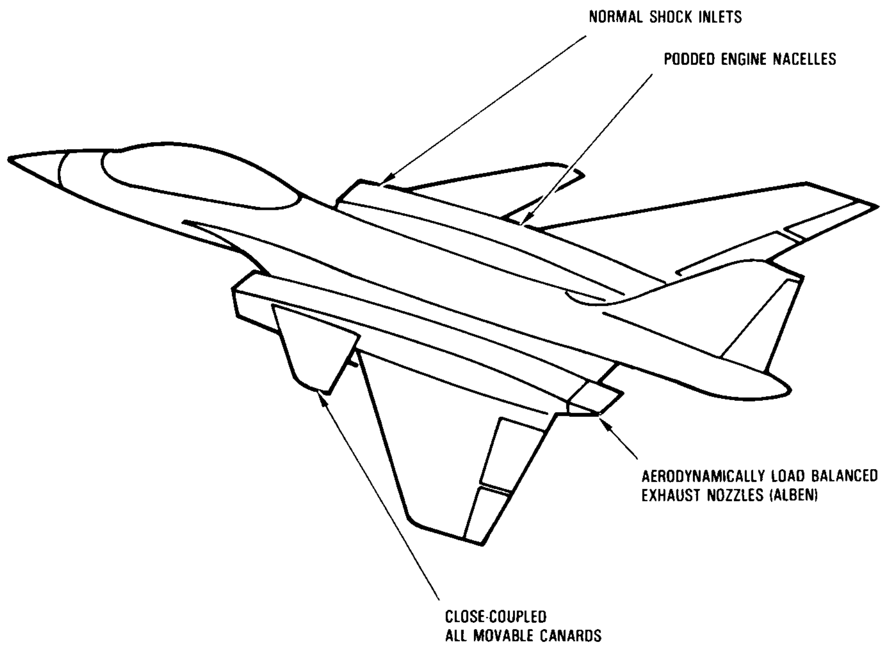

Figure 9 Supersonic V/STOL Advanced Nozzle Concept Fighter Configuration.

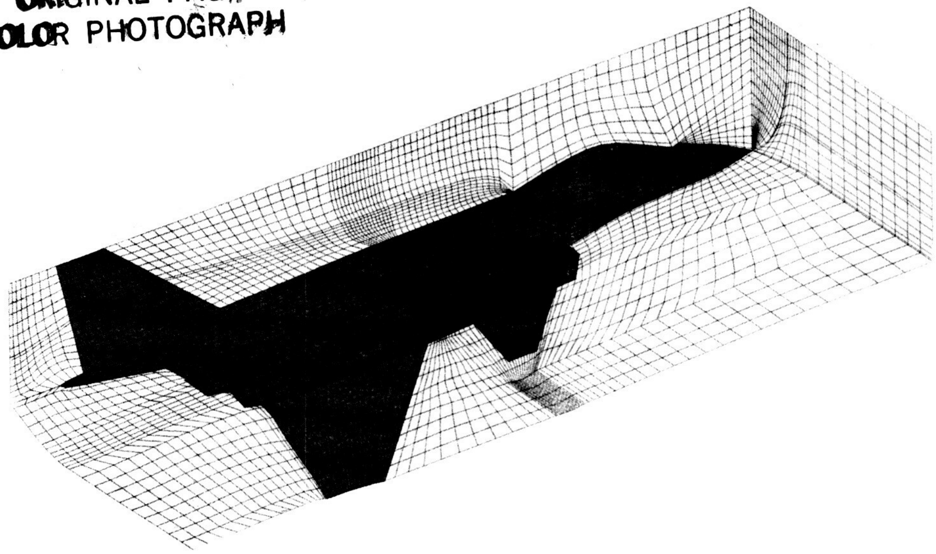

Figure 10 Partial View of Grid about ANC Configuration for Euler Computation.

Figure 11 Surface Pressure Distribution Computed by the TEAM Code (Euler Solution) at Angle of Attack = 4.8 Degrees and Mach Number = 1.2 - Upper Rear Quarter View.

ORIGINAL PAGE
COLOR PHOTOGRAPH

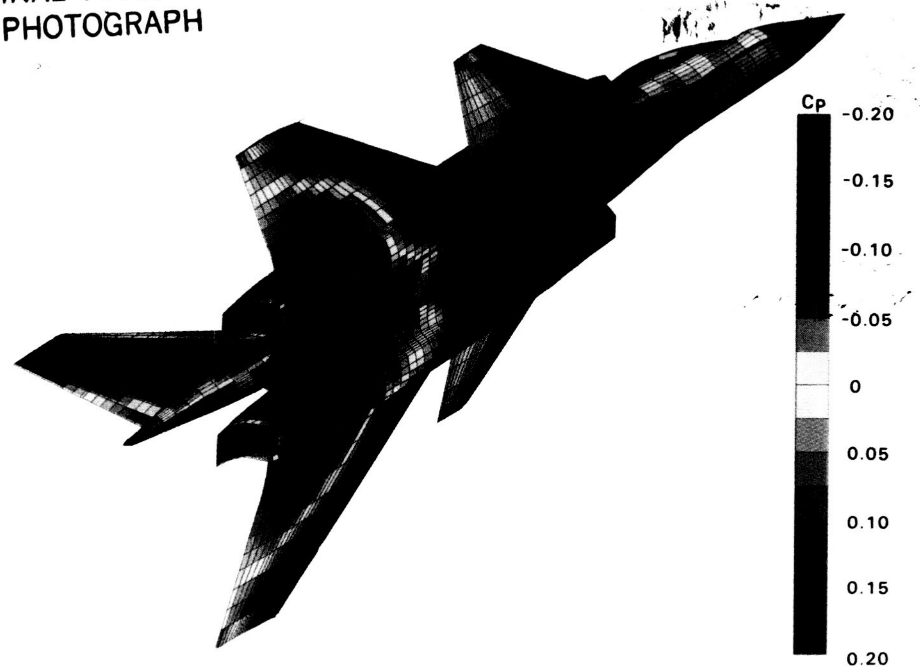

Figure 12 Surface Pressure Distribution Computed by the TEAM Code (Euler Solution) at Angle of Attack = 4.8 Degrees and Mach Number = 1.2 - Lower Rear Quarter View.

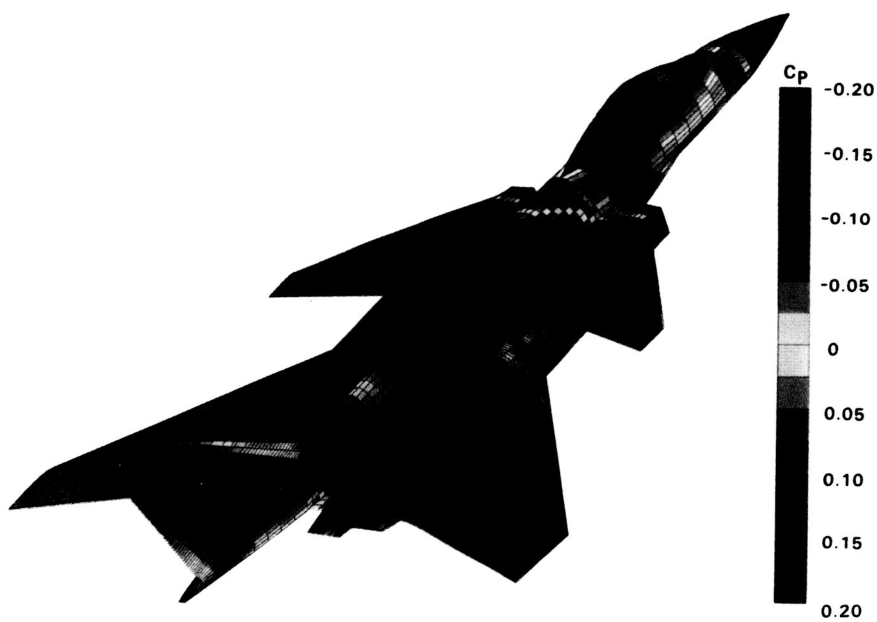

Figure 13 Surface Pressure Distribution Computed by the TRANSAM Code (Navier-Stokes Solution) at Angle of Attack = 4.8 Degrees, Mach Number = 1.2, and Reynolds Number = 6.5 Million - Upper Rear Quarter View.

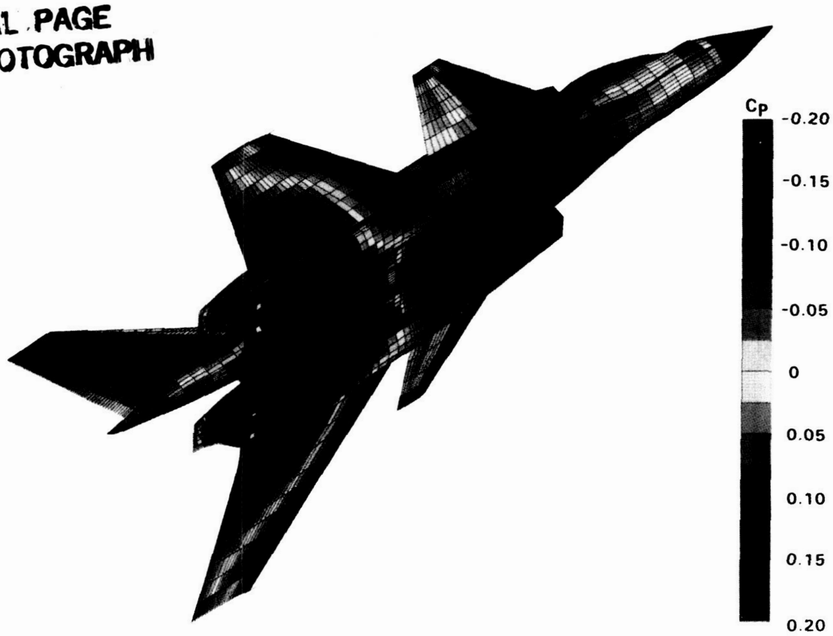

Figure 14 Surface Pressure Distribution Computed by the TRANSAM Code (Navier-Stokes Solution) at Angle of Attack = 4.8 Degrees, Mach Number = 1.2, and Reynolds Number = 6.5 Million - Lower Rear Quarter View.

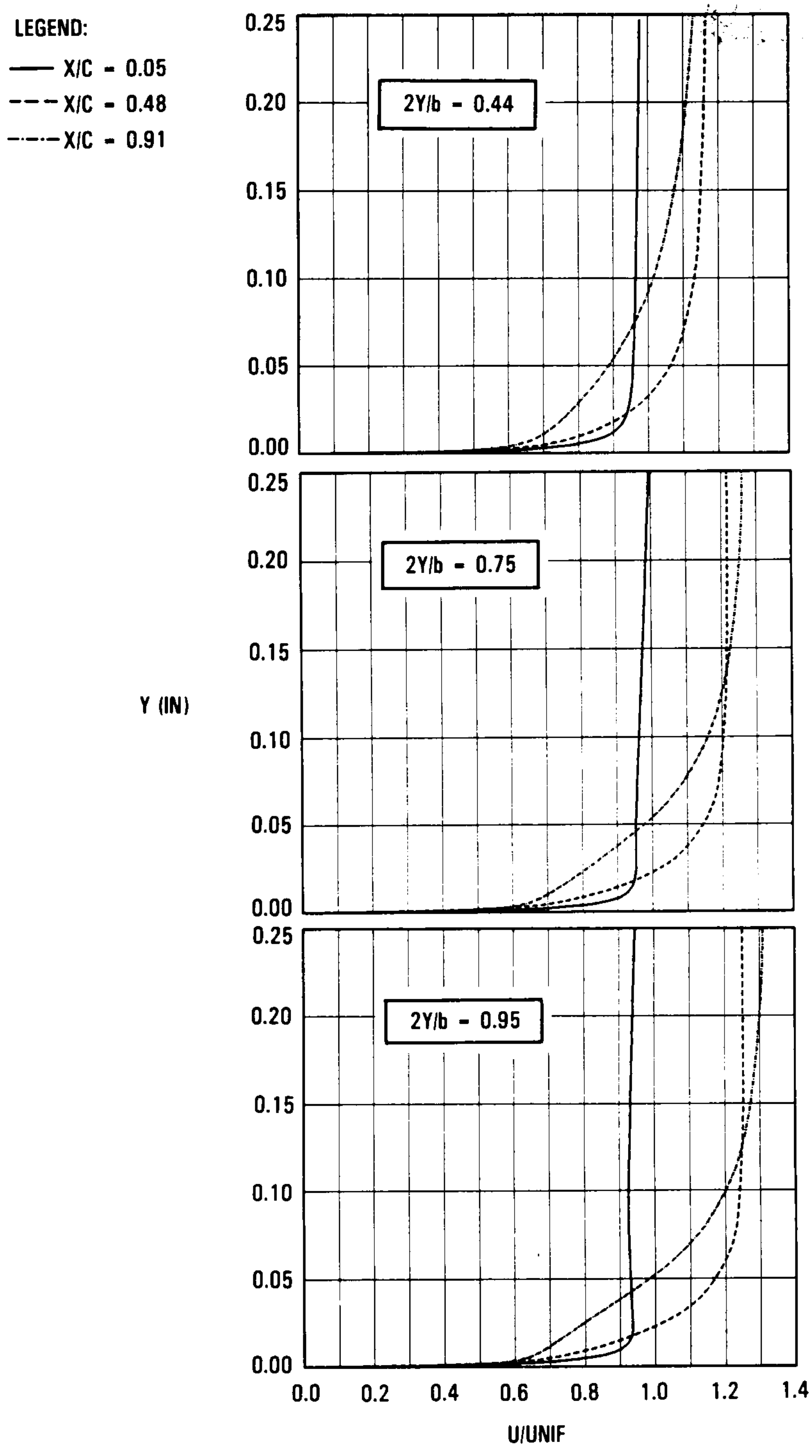

Figure 15 Wing Upper Surface Boundary Layer Velocity Profiles Computed by TRANSAM at Angle of Attack = 4.8 Degrees, Mach Number = 1.2, and Reynolds Number = 6.5 Million.

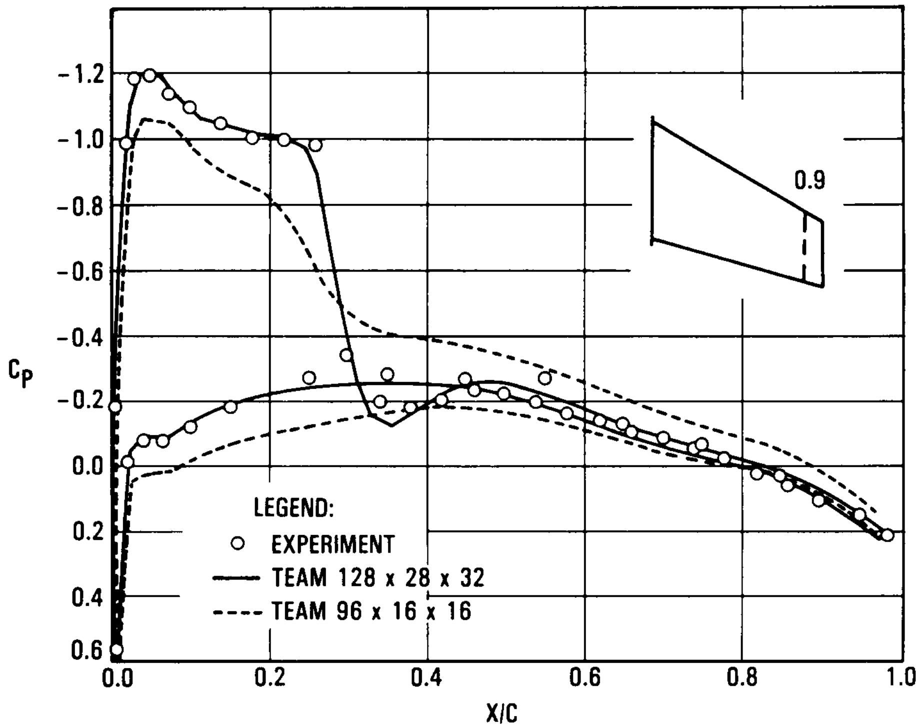

Figure 16 Effect of Grid Density on Euler Computation of Surface Pressure Distribution on the Onera M6 Wing Near the Tip. Mach = 0.84, and Angle of Attack = 3.08 Degrees.

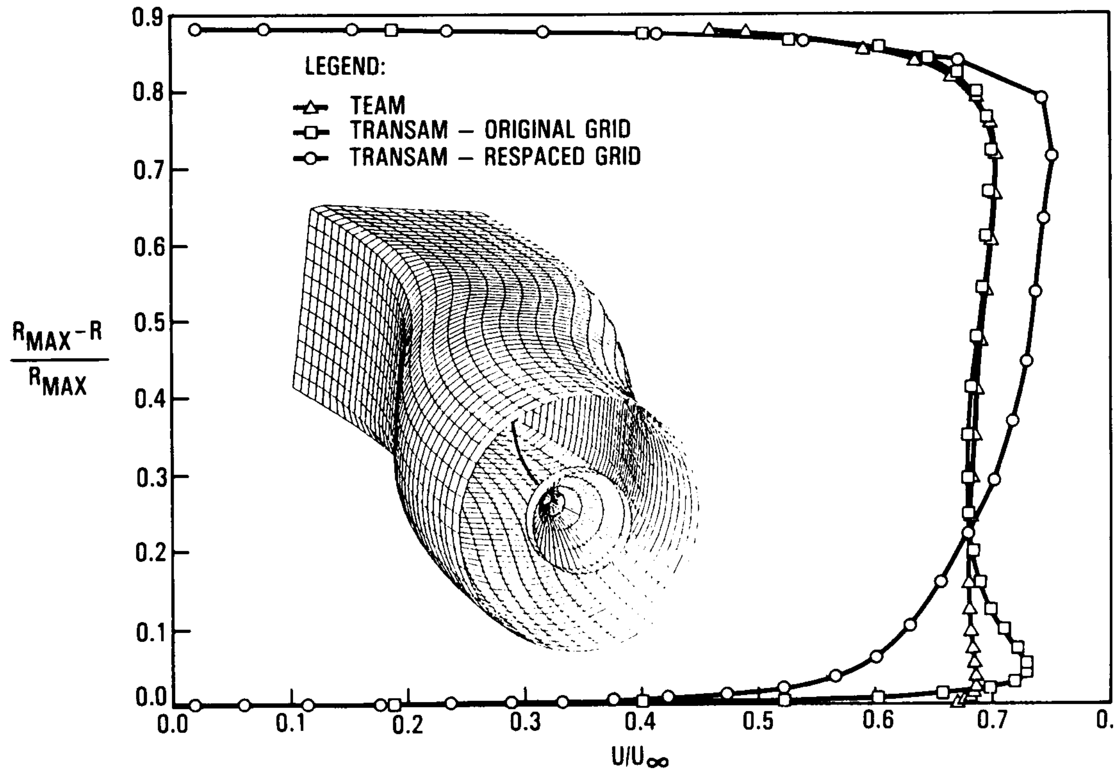

Figure 17 Computation of Internal Flow in a Subsonic Diffuser: Effect on Grid Point Distribution on Navier-Stokes Solution. Inlet Mach Number = 0.72.

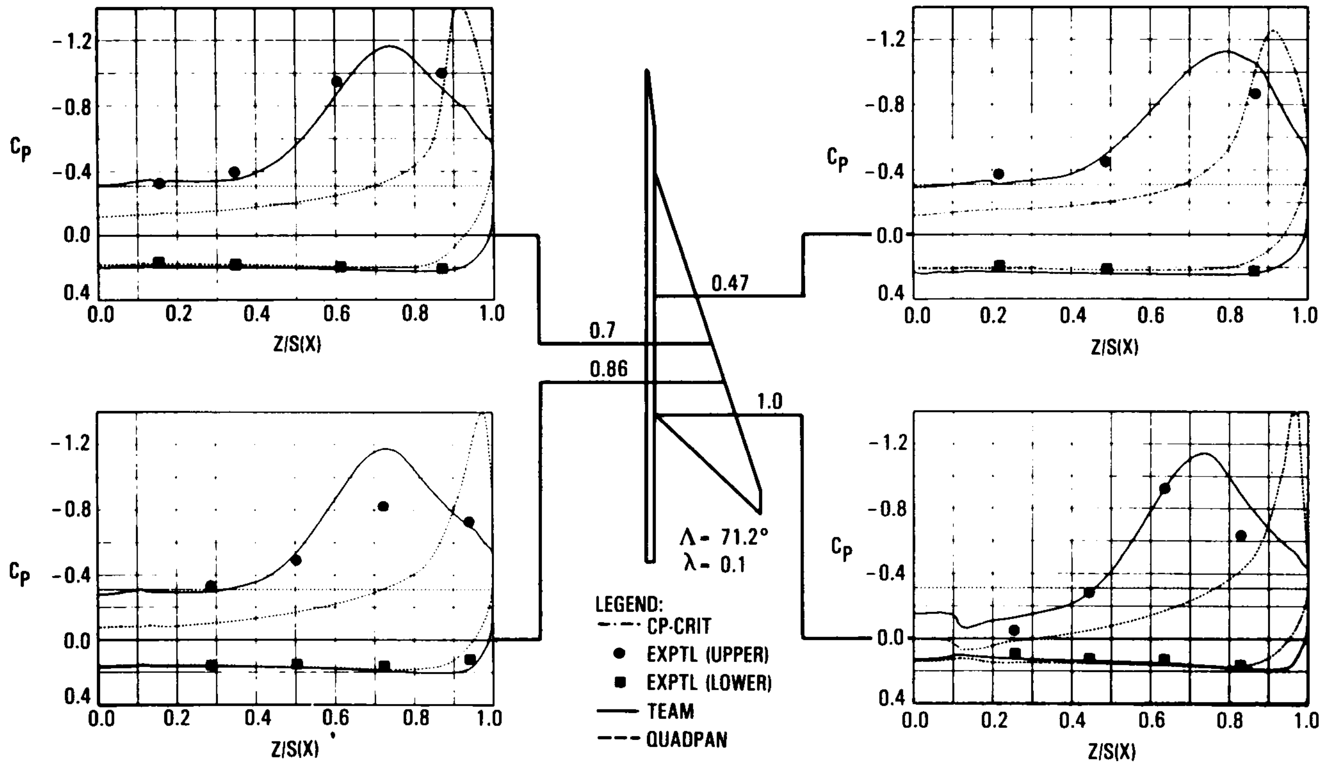

Figure 18 Cross-plane Pressure Correlation for Arrow Wing-body Configuration at Mach = 0.85 and Angle of Attack = 15.8 Degrees.