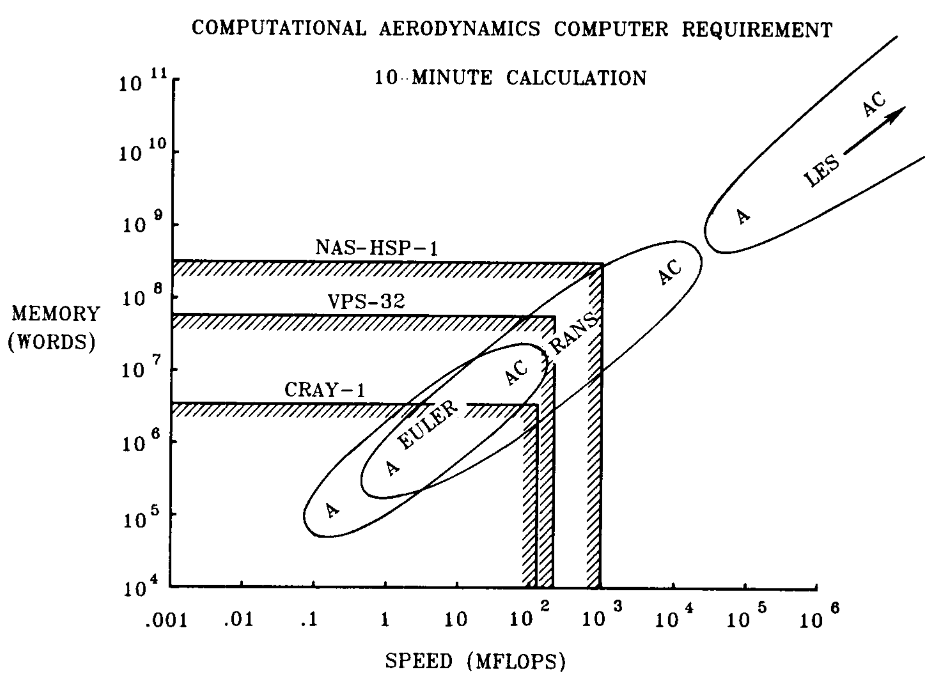
COMPUTATIONAL FLUID DYNAMICS OVERVIEW

Percy J. Bobbitt
Langley Research Center
Hampton, Virginia

[redacted] Computers and CFD

Our ability to treat more complex geometries with more complex or complete governing equations is governed by the speed and storage of our computers and the speed, or efficiency, of our solution algorithms. About a year ago, Langley's CYBER 203 was replaced by the VPS-32 with the potential for roughly an order of magnitude increase in memory and a slight increase in speed. Euler and Reynolds Averaged Navier-Stokes (RANS) solutions that can be expedited in approximately 10 minutes with this computer are depicted in the figure. Also, the capabilities of the NAS-HSP-1 (CRAY-2) and CRAY-1 are shown for reference. The A, AC, and LES symbols within the ovals indicate airfoil, complete-aircraft-configuration, and large-eddy-simulation calculations, respectively.

Clearly, we are at a point where the calculation of flows about complete aircraft (AC) using the Euler equations has become possible. (A recent AIAA paper by J. C. South, Jr.[1] indicates that the Euler boundary for complete aircraft (AC) calculation is on the optimistic side and that these types of calculations will probably require a NAS-like computer to achieve a 10-minute calculation time.) Two-dimensional as well as some three-dimensional flows can now be computed using the RANS equations. Storage is available for fairly complex geometries using RANS equations but computation time is excessive. The need for faster solution algorithms for those types of equations is clear. In the next few years, it is expected that computers with an order of magnitude increase in speed beyond the CRAY-2 will become available thus allowing "practical" RANS computations in a reasonably short time. With the present rate of growth of our computer technology, large eddy simulations of the flow about an airfoil will become possible in the early 1990's.



Potential Equation Research

A considerable amount of research is still being devoted to the various nonlinear forms of the potential equation. Included are activities aimed at improving the speed of computation, the ease of application to complex geometries, and the accuracy of the solutions. Many of the specific applications have necessitated significant improvements and modifications in existing codes. In some cases, a complete code development, including an appropriate grid system, was required such as that described in the first paper of the session by South et al. The second paper, by Salas and Gumbert, treats the nonuniqueness problem associated with the "conservative" formulation of the full potential equation and explains some of the large discrepancies encountered in the past between various airfoil codes. The application of a wing/body code due to V. J. Shankar to two supersonic fighter configurations is described in the paper by Jones and Talcott. Finally, one element of the analysis presented in the "vortex breakdown" paper by Luckring is based on the linear potential equation.

- MULTIGRID METHODS
- MASS FLUX BOUNDARY CONDITIONS
- WIND-TUNNEL FLOWS (+)
  - ASSESSMENT & CORRECTIONS
  - ADAPTIVE WALLS
- NONUNIQUENESS OF "CONSERVATIVE" EQUATION SOLUTIONS (+)
- WING/BODY APPLICATIONS (+) (*)
- VORTEX FLOWS (+)
- NOZZLE FLOWS
- PROPELLER SLIPSTREAM
- AIRFOIL DESIGN (*)

(+) PRESENTATION IN THEORETICAL AERODYNAMICS SECTIONS

(*) ILLUSTRATIVE RESULTS IN OVERVIEW

Euler Equation Research

It was noted in the discussion of our first figure that the VPS-32 computer has the capability of handling 3-D computations for the Euler as well as the RANS equations. The paper by Thomas et al. presents results from one such effort in which a second- or third-order accurate upwind scheme for the discretization of the convective and pressure derivatives (based on a technique developed by Van Leer) is used. A relaxation scheme for the unfactored, implicit, backward Euler time method is also applied. When treating RANS equations, central differencing has been used for the viscous terms. Results have been obtained for several 2-D problems for both the Euler and RANS equations; present activities are concentrated on extensions to 3-D.

Other theoretical research using the Euler equations includes wing/body, leading-edge vortex and transport-engine-nacelle flows. A code has also been developed to calculate the effects of a propeller slipstream on wing pressures. An example is shown later of an Euler wing/body calculation.

- FLUX SPLIT, UPWIND SCHEMES (+)
- WING-BODY APPLICATIONS (*)
- VORTEX FLOWS
- NACELLE FLOWS
- PROPELLER SLIPSTREAM

(+) PRESENTATION IN THEORETICAL AERODYNAMICS SECTIONS
(*) ILLUSTRATIVE RESULTS IN OVERVIEW

Navier-Stokes Equation Research

Computational fluid dynamics (CFD) research concerning the Navier-Stokes equations takes a variety of forms. The paper previously discussed by Thomas et al. presents an implicit upwind flux split scheme which leads to a diagonally dominant matrix structure, which allows large time steps to be taken in three-dimensional problems. In the paper by Swanson et al., a class of explicit multistage time-stepping schemes is used to construct a solution algorithm. Various methods are employed for accelerating convergence to steady state. Two-dimensional (airfoil) results have been obtained; three-dimensional programs are now being debugged.

Several codes have been written over the past few years which solve for the time-varying behavior of the viscous boundary-layer flow using the full Navier-Stokes equation. These solutions are normally referred to as numerical simulations. The paper by Zang and Hussaini presents the results of a numerical simulation of transitional flow and various methods for delaying transition. Another study of this type is given in a paper in the Fluid Physics Session by Maestrello et al.

Vortex flows are receiving a lot more attention in recent times from the CFD world. The paper by Weston et al. in this session presents results of several Navier-Stokes analyses in which the growth and decay of vortices over and behind a wing are simulated. Vortex breakdown or bursting is the subject of the Luckring paper. In his analysis, a viscous core and inviscid outer flow are matched to effect a consistent global solution.

In the last paper in this session, Kumar and Trexler treat the supersonic flow in a scramjet inlet utilizing the three-dimensional Navier-Stokes equations. Recent experimental results for two types of inlets are compared with predictions.

FLUX-SPLIT, UPWIND SCHEMES (+)

THIN LAYER NAVIER-STOKES - 2-D & 3-D (+)

TRANSITION SIMULATION - 2-D & 3-D (+)

VORTEX FLOWS (+)

SCRAMJET INLETS (+)

AFTERBODY/NOZZLE FLOWS (*)

(+) PRESENTATION IN THEORETICAL AERODYNAMICS SECTIONS
(*) ILLUSTRATIVE RESULTS IN OVERVIEW

Applied CFD and Validation Research

The papers given in the Theoretical Aerodynamics Session comprise a representative sample of the ongoing basic and applied CFD research programs. Time does not permit a synopsis of all of the research projects in progress; basic methodology investigations are particularly difficult to follow when "abbreviated". On the other hand, the flavor of a validation or applied activity can usually be imparted with just a few figures. Consequently the remainder of the overview will be aimed at giving a more complete picture of our broad-based applied CFD and validation programs. The applied CFD research has a number of goals including

- Code validation
- Increased understanding of flow physics
- Experiment definition
- Development of design procedures

The ATAT program, described below, and the other three activities chosen for review all have these goals but with varying emphasis.

Advanced Technology Airfoil Test (ATAT) Program

A cooperative program with the U. S. transport industry has just been concluded which was aimed at validating and improving airfoil design methodology at flight Reynolds numbers. An extensive series of correlation and advanced technology airfoils were tested in the Langley 0.3-Meter Transonic Cryogenic Tunnel; they are listed on the opposite figure. The design constraints were that the thickness ratio be 0.12, the c_ℓ be approximately 0.65 and the Mach number be 0.765. For the most part, the Korn-Garabedian code was employed by the participants, but some use was made of the Grumfoil code of R. Melnik. Criteria used for transition strips and sidewall suction were examined in several of the tests. Tunnel-wall interference corrections accounting for sidewall interference, top and bottom wall interference, or all four walls were also studied. Similar programs have been developed with industry for transport and fighter configurations with the NTF as a focus.

ADVANCED TECHNOLOGY AIRFOIL TEST (ATAT) PROGRAM

INDUSTRY

BCAC 1

BCAC 2

DAC

LAC 1

LAC 2

LAC 3

CORRELATION

NACA 0012

NACA 65-213

NASA SC(2)0510

NASA SC(2)0714

DFVLR CAST 10

DFVLR CAST 10, c/2

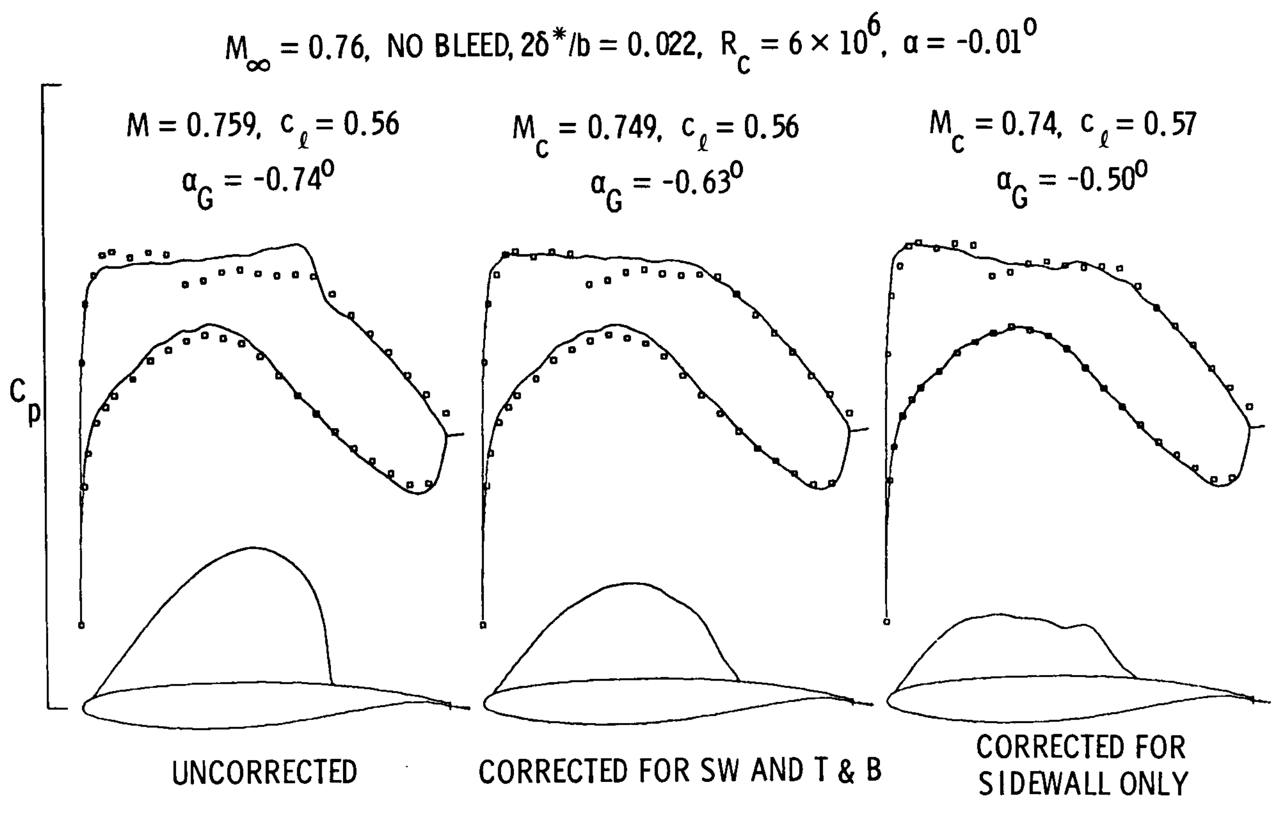
ADVANCED NASA

NASA SC(3)0712A

NASA SC(3)0712B

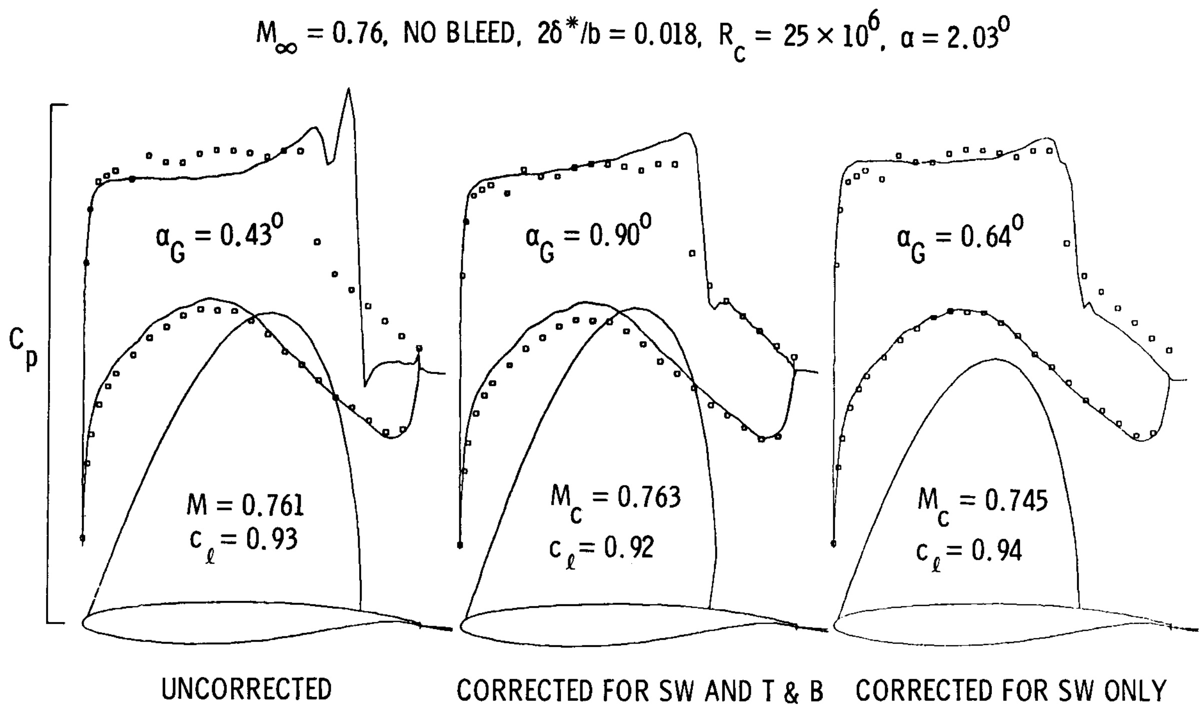
Comparison of Data With Grumfoil at Near Design Lift

The data shown in the figure below were obtained during the course of the ATAT program on the NASA SC(3)0712 airfoil. It was part of an investigation of the effects of sidewall bleed and the utility of various tunnel-wall correction techniques. Experimental results are for a normal tunnel Mach number of 0.76, a R_c of 6×10^6, and a c_ℓ of 0.56. The left-hand figure is a comparison of the data with the Grumfoil code run at the experimental c_ℓ and Mach number and shows that the theory yields too high a suction pressure level and a stronger shock on the top side than experiment. If the data are corrected for the sidewalls and the top and bottom walls (center plot), the corrected Mach number is lower and the geometric angle of attack (α_G) is slightly higher. The resulting pressures are in better agreement on the top with the bottom remaining essentially unchanged. Finally, if only the sidewall correction is applied, the agreement on both the top and bottom surfaces is improved over the four-wall-correction results. Clearly, the validation of airfoil design procedures requires a good understanding of tunnel-wall interference.



Comparison of Data With Grumfoil for a High-Lift Case

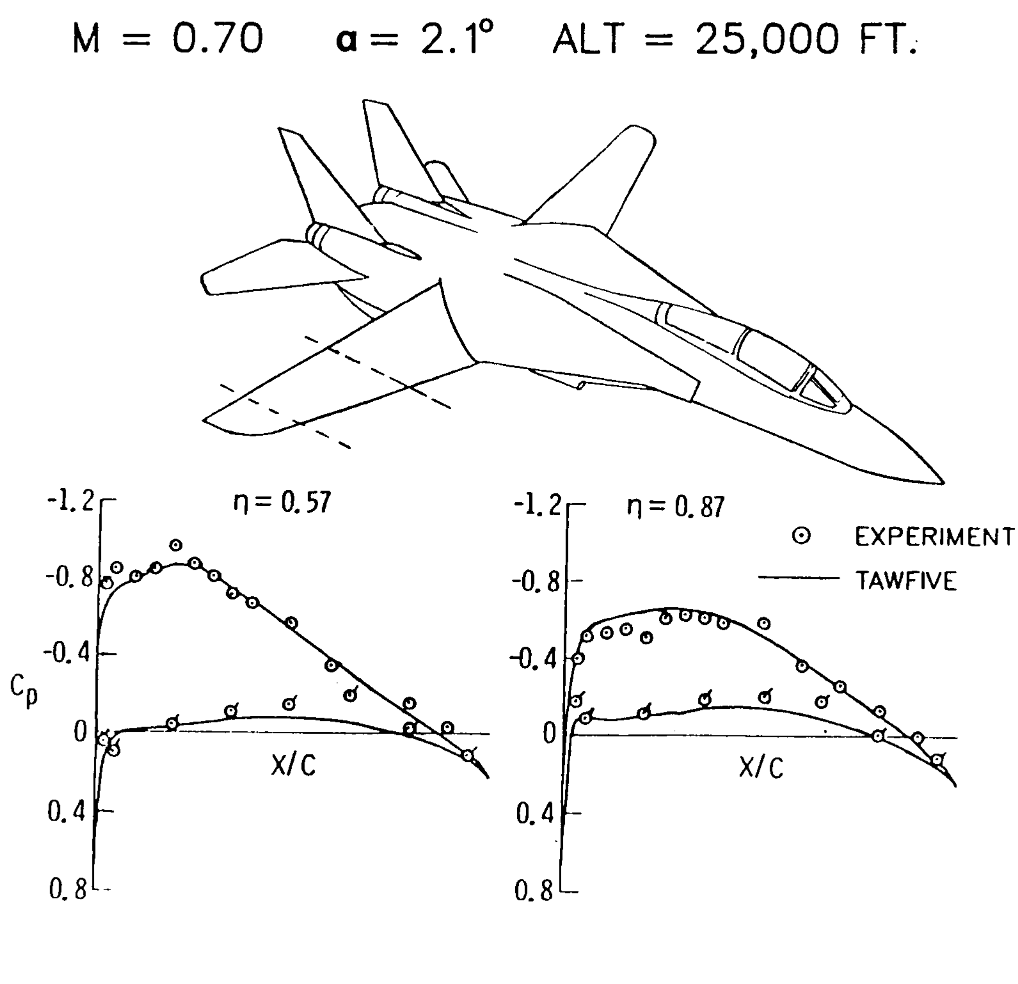
This is a plot similar to the previous one with the only differences being a higher Reynolds number (25×10^6) and a higher c_ℓ (≈ 0.93). The conclusion relative to the wall corrections is the same, i.e., the sidewall correction, by itself, provides the best agreement with experiment from the Langley 0.3-Meter Transonic Cryogenic Tunnel.



F-14 Variable Sweep Transition Flight Experiment

A number of transonic wing/body codes have been developed over the past half dozen years based on various forms of the nonlinear potential equation. One of the most accurate we have found is one developed by A. Jameson called FLO30. It has been mated by Craig Streett of the LaRC with a 3-D boundary layer routine based on the method of P. D. Smith to form the TAWFIVE code. In addition, he has "patched in" the strong trailing-edge interaction developed by R. Melnik of Grumman to provide a better accounting of the pressure variations normal to the surface and wake and the effects of wake curvature.

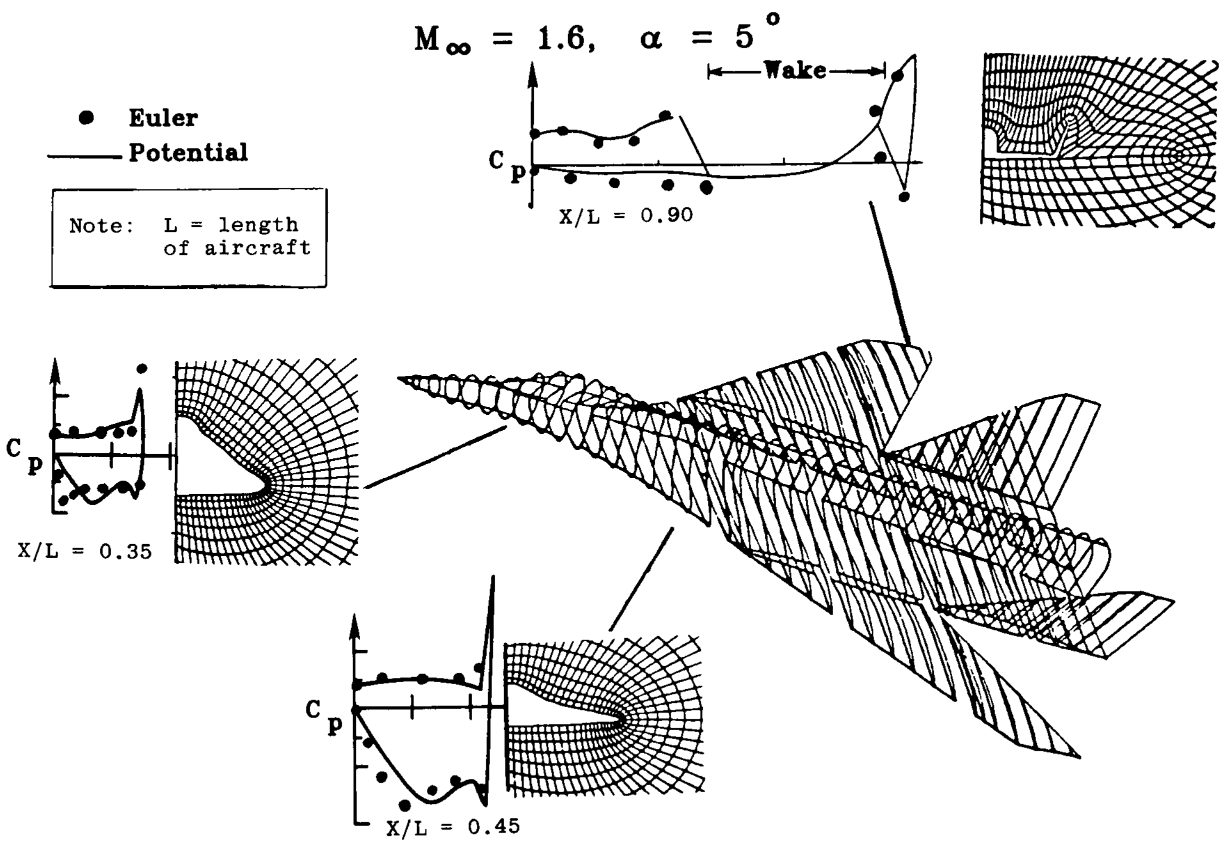
In the figure, the result of applying TAWFIVE code to the basic F-14 wing at two span stations is depicted. The agreement here is typical; many cases show even better agreement. More details of this application and others are given in a paper by Waggoner et al. in the Viscous Drag Reduction Session.



Euler Solutions for High-Speed Flow About Complex Three-Dimensional Configurations

High-speed flow about complex aerospace configurations has been simulated by numerically solving the Euler equations[2]. A finite-volume explicit scheme with Runge-Kutta time integration is employed to solve the three-dimensional compressible Euler equations. The incorporation of carefully chosen dissipative terms and convergence accelerations such as enthalpy damping and maximum time-stepping has rendered the method very efficient in solving high-speed flows involving strong shocks and local subsonic regions. Discretization of the computational space is achieved by an algebraic method developed for quasi-three-dimensional grid generation for blended wing-body geometries and other complex configurations. The method has proven its versatility as applied to realistic aircraft geometries. The software for both grid generation and flow simulation reside on the VPS-32 system.

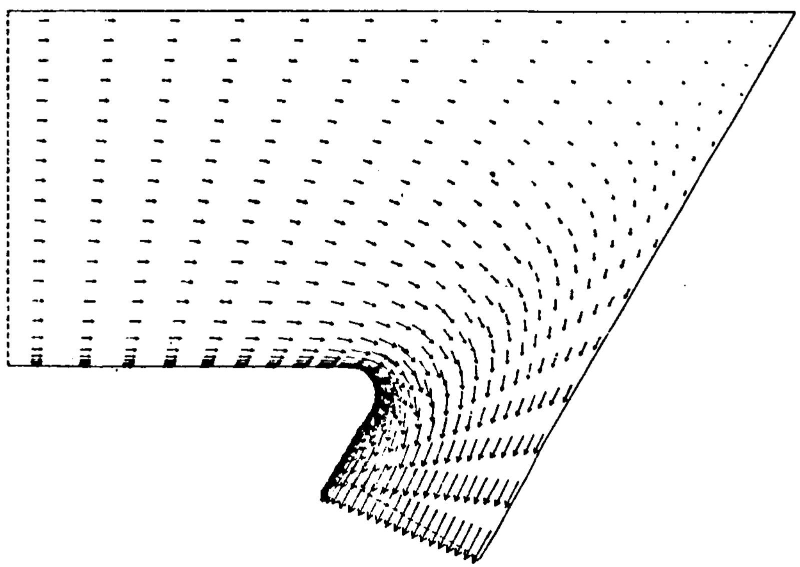
A typical set of results is presented in the figure below on this page. The body-geometry investigated is a fighter-like configuration used for full potential flow computations at the Rockwell International Science Center. Surface pressure plots are presented for three cross-sectional planes located along the axis of the body. A comparison of the full potential and Euler solutions presented shows a reasonable degree of agreement between the pressure values predicted by the two methods.



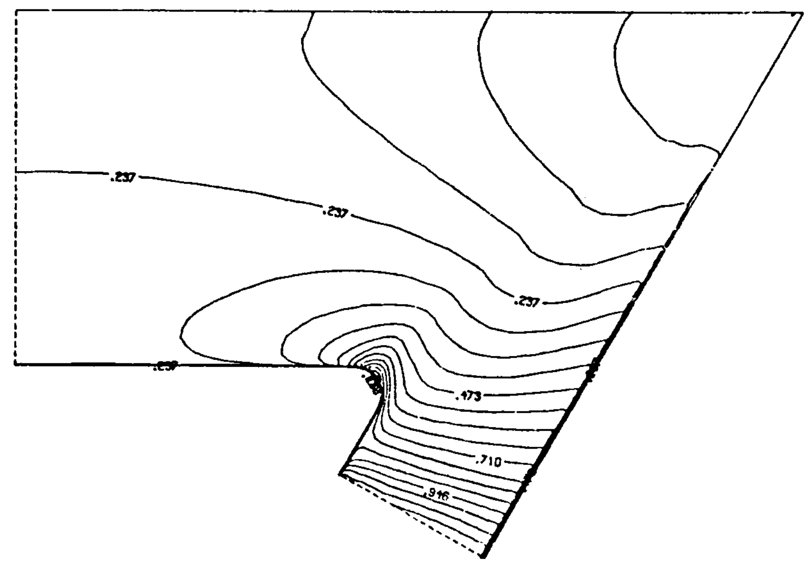
Navier-Stokes Results for Fully Deployed 2-D Thrust Reverser

A Navier-Stokes code has been developed by R. MacCormack at the University of Washington for solving the flow field in two-dimensional nozzles having up to three exhaust ports. The code solves the finite-volume form of the Navier-Stokes equations using MacCormack's explicit-implicit scheme combined with flux-vector splitting. Turbulence is modeled with the Baldwin-Lomax eddy viscosity model. The code has been applied to solving the flow in fully deployed thrust reverser ports such as that shown in the figure. The results are in good qualitative agreement with experiment for the limited cases calculated to date. A more detailed assessment of the accuracy of the computational model is under way.

NAVIER-STOKES RESULTS FOR FULLY-DEPLOYED 2-D THRUST REVERSER



VELOCITY VECTORS

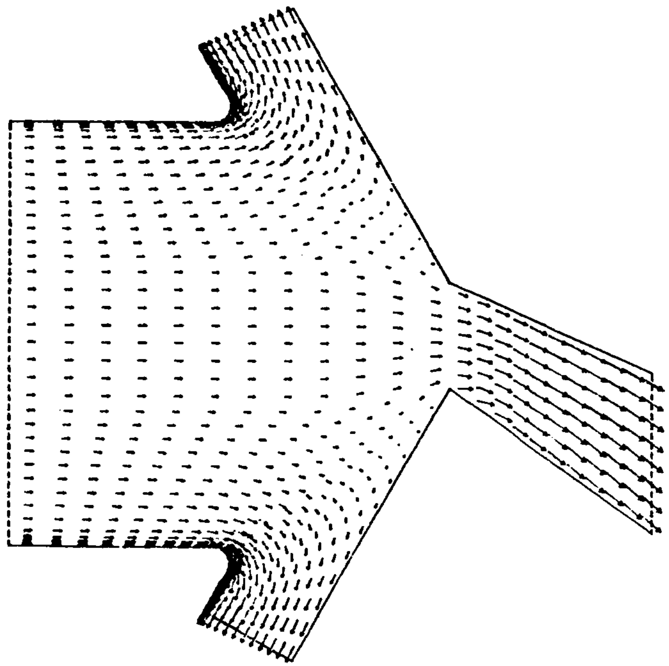


MACH NUMBER CONTOURS

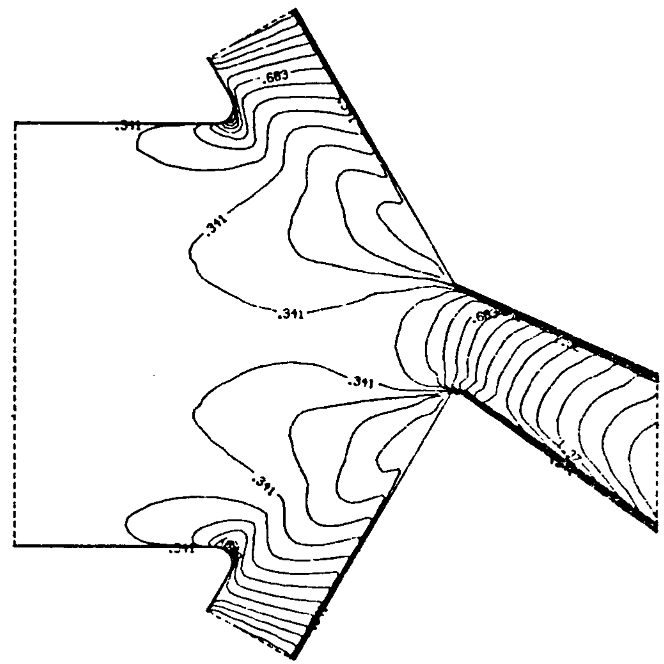
Navier-Stokes Results for Partially Deployed 2-D Thrust Reverser With Vectoring

The MacCormack 2-D nozzle code has also been used to calculate flows in nozzles with both thrust reversing and vectoring. Results from several test problems show the flow in the reverser ports to be relatively insensitive to the degree of vectoring. Comparisons with recent experimental data for more realistic geometries are currently being made by Green and Wilmoth, NASA Langley, and Imlay, University of Washington.

NAVIER-STOKES RESULTS FOR PARTIALLY-DEPLOYED 2-D THRUST REVERSER W/VECTORING



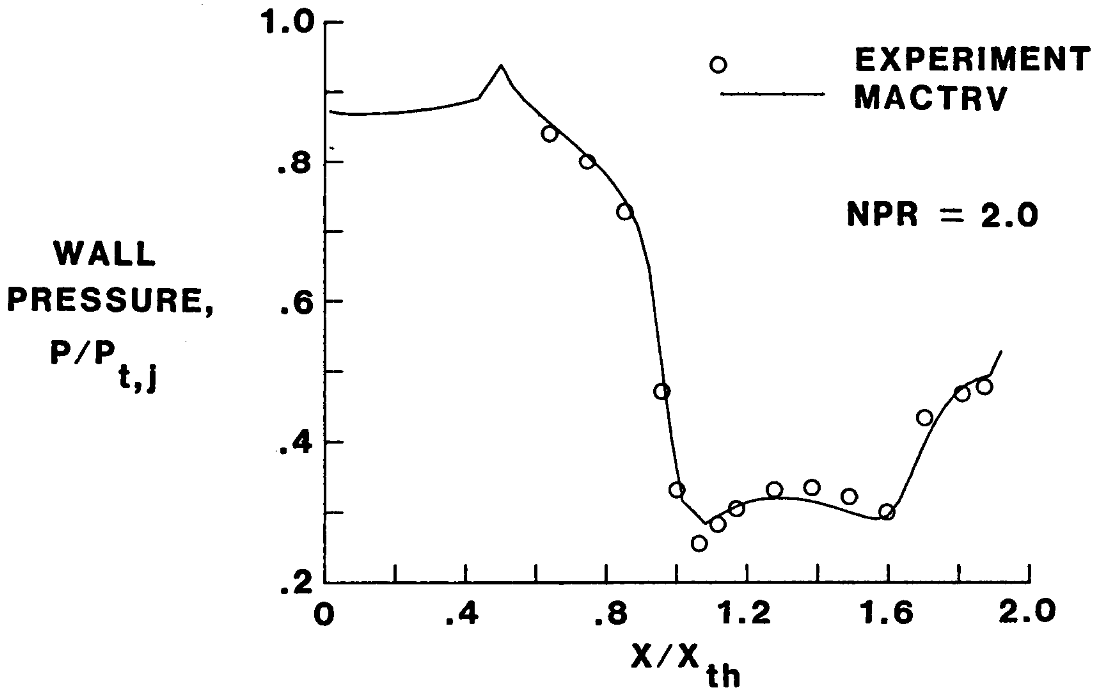
VELOCITY VECTORS

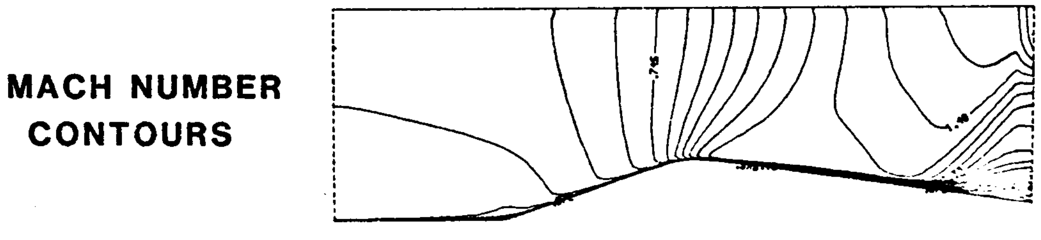


MACH NUMBER CONTOURS

Navier-Stokes Results for 2-D/C-D Nozzle

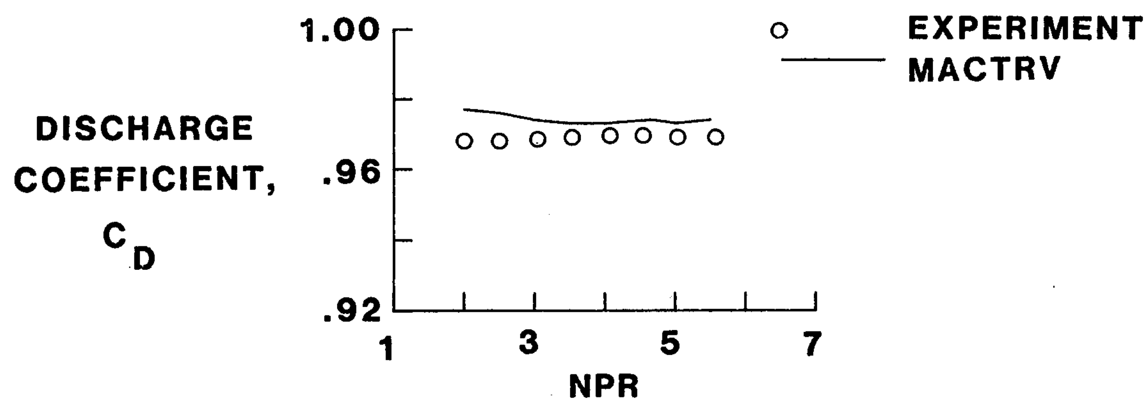
Comparisons of Navier-Stokes results (denoted by MACTRV) with experimental data of Re and Leavitt[3] for a 2-D/C-D nozzle have been recently obtained by Wilmoth. The agreement between wall pressures predicted by the 2-D code and experimental data along the centerplane wall is good over a wide range of nozzle pressure ratios. For the case shown (NPR = 2), good agreement is obtained over the entire nozzle length including the separated flow caused by nozzle overexpansion. These results give validity to the use of a 2-D approximation for rectangular nozzles with straight sidewalls.

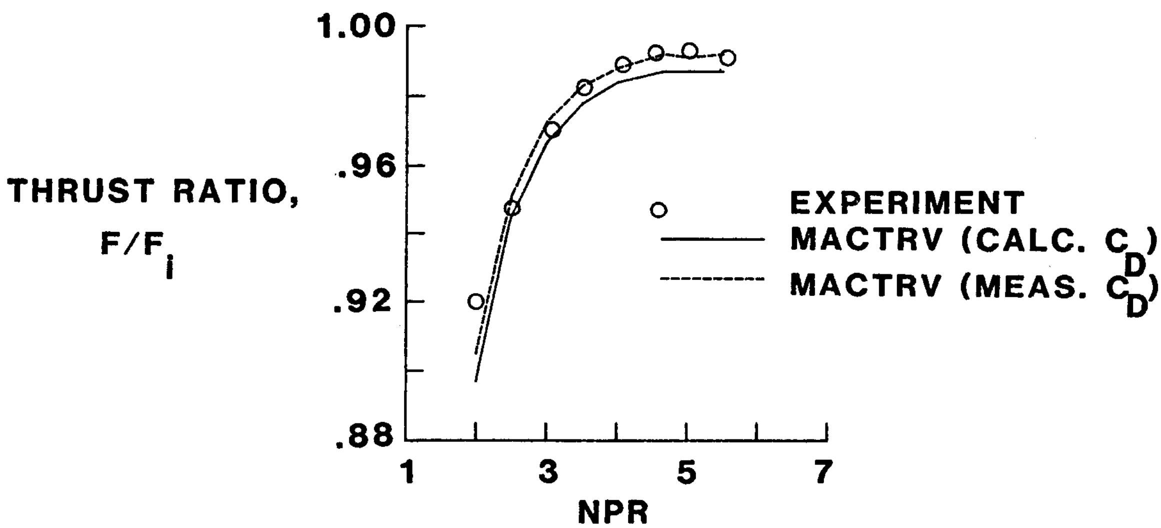




Navier-Stokes Results for 2-D/C-D Nozzle

Navier-Stokes predictions of 2-D/C-D nozzle performance parameters have also been made. For the case shown, both the discharge coefficient and thrust ratio agree with experiment to within 1 percent over a pressure ratio range of 2 to slightly over the design NPR. Use of the measured discharge coefficient to compute ideal thrust gives even better agreement for the thrust ratio. Similar studies for nozzles with thrust reversing and vectoring are planned.





Concluding Observations

The bulleted items on the figure represent a collection of opinions about what is important and what are the areas that need more work. The first bullet derives from observing over several decades the way industry, as well as some individual applied-code researchers, operates. Rarely does one doing applied work want to pay the cost of obtaining the world's most precise answer. Indeed he doesn't need it very often. Codes able to give reliable trends will always have a place in research and design. Long-running, high-cost codes are most often used for "vernier" adjustments within the circle of final design space or for clarification of a particular flow phenomenon.

The uniqueness problem outlined in the paper by Salas and Gumbert may exist for governing equations other than full potential and for 3-D problems as well. Clearly it is necessary to be particularly careful when conservative formulations are used for flow fields with shocks. Difference schemes resulting in artificial viscosity which is significant in magnitude compared with the natural viscosity are another concern. Grid refinements will frequently reduce or illuminate the problem. Obviously, stable solution schemes which have little or no residual artificial viscosity are to be preferred.

Computing equipment, like the family budget, is immediately saturated following an increase in capacity. It is desirable then to increase the efficiency of our solution algorithms, so that more computing can be accomplished with a given system or, perhaps of equal importance, simply to reduce the cost of a given computation.

With the activation of computers which are capable of Navier-Stokes calculations including simulations, the amount of information computed and available (and sometimes required) is becoming mind boggling. More and more use of graphics in our debugging strategies and data analyses is clearly needed. Carpet plots and "isolines" will be in demand. In any case the large data bases created must be accessible in "real time" and at the work site.

The trend of our experimental research is toward more basic phenomena all the time. Hardly anyone is satisfied with simply measuring forces and moments. Surface pressure data, flow field velocity vectors of both the steady and unsteady variety, gas composition, and transition boundaries are just a few illustrations of the quantities which are now being measured to validate our theoretical predictions. If possible we make the measurement using nonintrusive techniques. The money and time required to carry out a comprehensive experiment are often measured in hundreds of thousands of dollars and years, respectively. Highly skilled researchers are required and are in short supply. We are much more in need of good diagnosticians than CFD practitioners, and there is certainly no surplus of the latter.

CONCLUDING OBSERVATIONS

- HIERARCHY OF CODES ALWAYS NEEDED
    - BOUNDARY LAYERS & TURBULENCE MODELS IMPORTANT FOR FORESEEABLE FUTURE
- UNIQUENESS & ARTIFICIAL VISCOSITY ARE STILL CONCERNS
- EFFICIENCY OF SOLUTION ALGORITHM IMPORTANT NO MATTER HOW FAST OUR COMPUTERS GET
- DATA DISPLAY & ANALYSIS BECOMING MORE OF A PROBLEM
- EXPERIMENTAL VALIDATION RESEARCH BECOMING MORE BASIC, DIFFICULT & COSTLY

References

1. South, J. C., Jr.: Recent Advances in Computational Transonic Aerodynamics. AIAA Paper 85-0366, January 1985.

2. Moitra, A.: Numerical Solution of the Euler Equations for High-Speed, Blended Wing-Body Configurations. AIAA Paper No. 85-0123, 1985.

3. Re, Richard J.; and Leavitt, Laurence D.: Static Internal Performance Including Thrust Vectoring and Reversing of Two-Dimensional Convergent-Divergent Nozzles. NASA TP-2253, 1984.