TRANSONIC PROPULSION SYSTEM INTEGRATION ANALYSIS
AT MCDONNELL AIRCRAFT COMPANY

Raymond R. Cosner
Computational Fluid Dynamics
Propulsion/Thermodynamics Dept.
McDonnell Aircraft Company
McDonnell Douglas
St. Louis, Missouri

INTRODUCTION

The technology of Computational Fluid Dynamics (CFD) is becoming an important tool in the development of aircraft propulsion systems. This is largely because CFD analysis methods are gaining the versatility and reliability needed for engineering applications. Two of the most valuable features of CFD are:

- Acquisition of flowfield data in days rather than months.

- Complete description of flowfields, allowing detailed investigation of interactions.

Current analysis methods complement wind tunnel testing by:

- Screening proposed test parametrics, increasing the productivity of wind tunnel programs.

- Assisting in determining the type and location of wind tunnel instrumentation.

- Providing means to assess wind tunnel wall and support interference.

- Helping to interpret wind tunnel data and analyze problems.

This discussion is focused on CFD methods. However, aircraft design studies need data from both CFD analysis and wind tunnel testing. Each approach complements the other.

NOMENCLATURE

BL	Buttline
CAD	Computer-Aided Design
CFD	Computational Fluid Dynamics
Cp	Pressure Coefficient $(P-P_\infty)/q_\infty$
FS	Fuselage Station
MCAIR	McDonnell Aircraft Company
P	Pressure
q	Dynamic pressure
WL	Waterline
α	Angle of Attack
β	Angle of Sideslip, Yaw

Subscripts

t	Total (stagnation) state
∞	Freestream state

Several characteristics of an ideal engineering CFD capability can be identified:

- A set of grid generation and flowfield prediction codes applicable to the full range of geometry and flow conditions.

- A database of validated predictions, compared with wind tunnel or flight data, demonstrating that the attainable results are satisfactory for engineering studies and how the results should be adjusted (if appropriate).

- An experience base, defining the steps to be followed in obtaining best results.

- An affordable cost of obtaining solutions -- in terms of both computing resources and engineering labor.

- A system that is easy to learn.

- Rapid turnaround of geometric or flowfield parametric variations.

- Suitable tools to assist the user in preparing and verifying geometric and flowfield input data.

- Suitable tools to assist the user in examining results, and extracting engineering data from CFD results.

- Communication interfaces to reformulate CFD results for input to other codes (e.g., structural, loads, or thermal analyses).

Omission of any of these characteristics can reduce the usefulness of a CFD system for engineering analysis. These items are a statement of objectives rather than a description of our current methods. Improvement is needed. Our efforts have been guided by several general policies. We have chosen to:

- Develop methods to be used by engineers without special training in CFD.

- Develop a limited number of multi-purpose codes, rather than a large number of specialized codes.

- Use a common database format for CFD results.

These objectives represent our interpretation of the best manner to inject CFD into the engineering environment. These goals will be discussed later in this section.

Most engineers and scientists who develop CFD methods are amazed at the reductions in computing costs which have occurred over the past decade, and are continuing to occur. At the same time, however, there has been a sharp increase in the quantity and complexity of CFD analyses. The computer time per job which marks the limit of engineering capability is, in our experience, on the order of 30 minutes to two hours on whatever computers are available. However, the capability that a CFD engineer considers attractive and inexpensive today is usually seen as extremely expensive by a project engineer.

A key consideration at MCAIR has been the need to streamline the operation of CFD codes to the point where non-CFD engineers can use these methods effectively with minimal support from specialists. This approach has, in a sense, been forced by the lack of adequate numbers of CFD specialists. Also, this situation has occurred because some classified projects require a dedicated engineering staff.

The application of new CFD capability passes through several evolutionary stages at MCAIR:

1) Initial development and validation, up to the point where the CFD engineer believes the code can be used for an engineering purpose by someone else.

2) Application by other technology engineers to problems of interest, with continuous support by the original code developer. In parallel, a more thorough validation is performed. The code is modified as a result of this initial experience.

3) After successful application has been demonstrated, engineers on various projects start using the new capabilities with consulting support from CFD specialists.

4) Eventually, project engineers develop enough experience and confidence that they need little support from the CFD specialist.

The key to effective engineering application is to build a high level of versatility into the CFD procedures. This is done in several ways.

At MCAIR, several different Computer-Aided Design (CAD) systems are used, depending on the application, to generate geometric data. Automated tools have been developed to access and translate these different geometric data systems. Improvement of these tools is continuing. The product of these interfaces is a common geometric data format for CFD analysis.

A 3D multiple zone mesh generator has been developed based on this standard geometry format. It serves several different flowfield codes.

The flowfield codes must be applicable to a range of configurations and should not require tuning of non-physical input parameters to obtain results. This goal, for example, favors upwind codes rather than central difference codes which require specification of artificial viscosity parameters or smoothing coefficients.

With a high versatility in each code, the total number of codes can be minimized. This leads to advantages in training engineers, and in maintaining and upgrading the CFD capability.

Common post-processing tools are used so that all CFD data are processed consistently. These tools are supported by a common CFD data file structure. With this common data structure, only a modest number of interface procedures are needed to communicate CFD results to other codes for additional processing.

The project engineer needs tools to assist in preparing CFD input data - both geometric and flowfield data. These include graphics tools to display and verify the input geometry prior to the expense of a 3D mesh generation job.

A set of validation comparisons with test data must be provided. These check cases should be extensive, since they will have maximum value if some of the cases are close to the desired engineering application.

PRE-PROCESSORS

CFD pre-processors assist the user in preparing and verifying input data for the grid and flowfield codes. They are used for:

- Accessing and reformatting data from CAD data bases.

- Displaying and verifying the geometric data.

- Modifying the geometric data.

- Preparing input data files.

Several different systems are used to generate and store geometric data at MCAIR. The most commonly used are the loft data base and the Unigraphics system. Loft data are accessed interactively, with the engineer selecting section cuts and individual points on a workstation. As the points are picked, the coordinates are written to a file. This file is reformatted for input to the CFD procedures. Efforts are in progress to eliminate the man-in-loop process by generating input data from a batch procedure. Batch setup procedures are already in use for Unigraphics data. Interactive interfaces have been prepared for the less commonly used geometric systems.

The product of these interface procedures is a standard geometric definition file. This is a formatted file; the geometry is defined by a sequence of cuts. Each cut is defined by a string of points. For simple geometries, the input data can be set up or modified manually, using a text editor.

Several features have been built into the geometry description method to improve user friendliness. Chief among these is an approach where each cut is defined independently of the other cuts. For example, successive cuts need not have the same number of points. The points within a cut, and the cuts themselves, need not be spaced uniformly. The geometry description is independent of the grid to be constructed.

An interactive procedure has been prepared to display the geometric data prior to executing the grid generator. This verification tool is hosted on a VAX computer and provides displays on Tektronix terminals. The user can examine all or part of the geometry from any viewing angle.

412

The geometric data file can be modified readily by the user. Several simplified procedures have been written to assist basic propulsion studies, such as changing the compression angles on an inlet ramp. Further efforts in this area are in progress.

GRID GENERATION

Grid generation is often the bottleneck in providing CFD support for engineering studies. One of our top priorities is to improve the application of CFD by developing improved grid generation methods. We need to generate grids quickly and reliably for a range of complex, realistic flight vehicle geometries.

Many approaches are available to meet these needs. We have chosen to develop the approach of dividing a complex solution domain into several sub-domains. Each of these sub-domains, or zones, is geometrically simple and spans a unique region of space. Grid and flowfield algorithms have been developed to provide continuous solutions across the non-physical boundaries between zones.

Two grid generators have been developed for 2D applications. The primary focus of these methods has been inlet configuration modeling, although nozzle applications also have been examined.

The INLETG grid generator was developed for a range of 2D inlet applications. This method has been used for all published applications of our FANSI inlet code, as discussed in the next section and also in References 1 and 2.

Recently, a new 2D grid generator, Program INOZG, has been developed. This method extends the multiple zone method to an arbitrary set of 2D bodies in a flowfield, with arbitrary inflow/outflow boundary conditions as appropriate. The primary advance in this new method is the modeling of multiple inlet and nozzle flow passages.

Three-dimensional multi-zone grids are provided by Program ZGRID. This code supports three full Navier-Stokes methods: X3D, NASTD, and CFL3D (single zones only). ZGRID is applicable to partial or full configurations (fuselage with optional addition of inlet, wings, and tails) and to selected components (isolated inlets, isolated ducts).

FLOWFIELD-PREDICTION CODES

A wide range of flowfield prediction codes are used in transonic propulsion integration studies at MCAIR. These range from 2D integral boundary layer methods to 3D zonal time-marching Navier-Stokes codes. Only codes based on the Navier-Stokes equations will be discussed in detail.

Velocity-Split Navier-Stokes Methods - Three analysis methods have been developed at MCAIR based on the velocity-splitting method. The first procedure (Ref. 3) was developed as a testbed for axisymmetric afterbody-plume combinations and is not an engineering tool. The second procedure, Program AFTEND, applies to analysis of 3D forebody-afterbody combinations. Several results from AFTEND have been published (Refs. 4, 5, and 6).

The same methodology was implemented in Program X3D, with improvements for multiple zone analysis and improved vector processing efficiency. This code uses our ZGRID grid generator. X3D is currently considered our most sophisticated engineering tool for 3D transonic viscous analysis.

Time-Dependent Navier-Stokes Methods - Two time-dependent analysis codes are in use or under development. Both are based on non-overlapping multi-zone grid methods to achieve maximum geometric capability.

A two-dimensional code, Program FANSI, was developed for inlet flowfield analysis (Refs. 1 and 2). This code has gained wide application at all speeds of interest, and is the most-used Navier-Stokes code at MCAIR.

A three-dimensional code using the same methodology as FANSI is under development. This program (NASTD) is an implicit, upwind, finite-volume code based on the approximate factorization method. NASTD also uses the ZGRID grid generator, and therefore has the same geometric applicability as Program X3D. Due to the higher solution cost of this time-marching methodology, NASTD is generally used on a limited basis to check results from X3D. At this time, the primary applications of NASTD are at higher speeds.

In summary, analysis of propulsion integration flowfields is based on a family of analysis methods of varying sophistication and cost. Several methods have been developed based on the Navier-Stokes equations. Results from these methods will be presented below. Initial inlet and nozzle trade studies often are based on the 2D Navier-Stokes code, FANSI. Transonic trade studies and integration analyses are performed by X3D, with selective use of NASTD.

POST-PROCESSING

Post-processing provides the tools for the project engineer to extract needed information from CFD results. To simplify post-processing, we have defined a Common Data File for CFD results. All codes used for propulsion analysis enter their results into this file structure for post-processing and for long-term storage. Currently, 26 CFD codes are integrated into this system. These include 2D (planar, axisymmetric) and 3D codes using Euler, PNS, and full Navier-Stokes methodologies. Codes using both perfect and real gas models are supported, along with combustion analysis programs.

This structure is a random-access file, storing the data in binary form. This format provides rapid input/output (I/O) and avoids the need to read and store a full flowfield file in order to use the post-processors. Binary-to-binary conversion routines have been prepared to transfer data between dissimilar computers. Currently, these common data files are created on Cray, VAX, and Convex computers; they are generally moved to a VAX computer for post-processing.

All propulsion post-processing packages access this Common Data File. Thus, our engineers only need to learn to use one set of graphics software. Full graphics support can be provided quickly for a newly received code simply by inserting interface subroutines that write the data into the common format. This interface task typically requires about one day. Another advantage is that a single set of post-processing software is used. Therefore, all CFD results are processed consistently regardless of their source.

At MCAIR, CFD graphics are supported using Tektronix terminals (both black/white and color displays) and the Silicon Graphics IRIS Workstation. Several software packages are in use:

PLTTR - This package, which was developed at MCAIR, is hosted on a VAX or Convex computer and presents displays on Tektronix 40xx and 41xx terminals. Several functions are available. The most commonly used are contour plots, vector plots, surface function plots (Cp vs. x, etc.) and boundary layer profiles. These plots are not restricted to grid surfaces; linear interpolation is performed as required. Experimental data and more than one set of CFD data can be presented on the same plot for comparison.

Other functions include a force/moment integration, printout of various properties, and integration of flow properties in an arbitrary closed contour in the flowfield.

MOVIE.BYU - This is a program developed at Brigham Young University for displaying 3D geometries and flowfield properties. At MCAIR, it operates on VAX computers with the displays presented on Tektronix terminals. An interactive interface program has been written to extract data from the Common Data File along selected grid surfaces for display by MOVIE.BYU.

PLOT3D - PLOT3D is a software package developed at NASA Ames Research Center which runs interactively on the Silicon Graphics IRIS Workstation. This system has a tremendous range of capabilities and in most respects is our most powerful graphics tool. We have written a menu driver for PLOT3D, to facilitate its use by inexperienced engineers. This package is described in Reference 7.

TRACE - This is a program developed at MCAIR to compute and display off-body or surface streamlines. It runs on a VAX computer, and provides displays on a Tektronix terminal. Recently, we have used the surface streamline capability to define flow properties along an Euler streamline at a wall, providing data for subsequent strip analysis using 2D boundary layer codes.

ARTIS - ARTIS is a software set developed at Douglas Aircraft Co. for interactive displays on the IRIS Workstation. Its display capabilities, which are discussed in Reference 8, are generally similar to MOVIE.BYU. ARTIS is a menu-driven package. Our engineers generally need less than 30 minutes of instruction to become self-sufficient. This is the most commonly used graphics package at MCAIR, for engineers who have access to an IRIS Workstation.

APPLICATIONS

Several representative applications of CFD in transonic propulsion system integration are discussed below. Many of these results have been published previously.

Forebody Flowfield - Accurate prediction of forebody flowfields is important to selecting candidate inlet concepts and integration locations. In particular, the engineer needs accurate data on flow angularity, local Mach number, and local total pressure recovery.

The flowfield for an isolated forebody, depicted in Figure 1 was computed with Program X3D. The flow condition is Mach 0.9, 10° angle of attack (α). This calculation, which was previously presented in Reference 4, was validated with experimental data acquired under MCAIR IRAD funding. The computed flow angularity is in good agreement with test data. Typically, the mismatch is less than one degree.
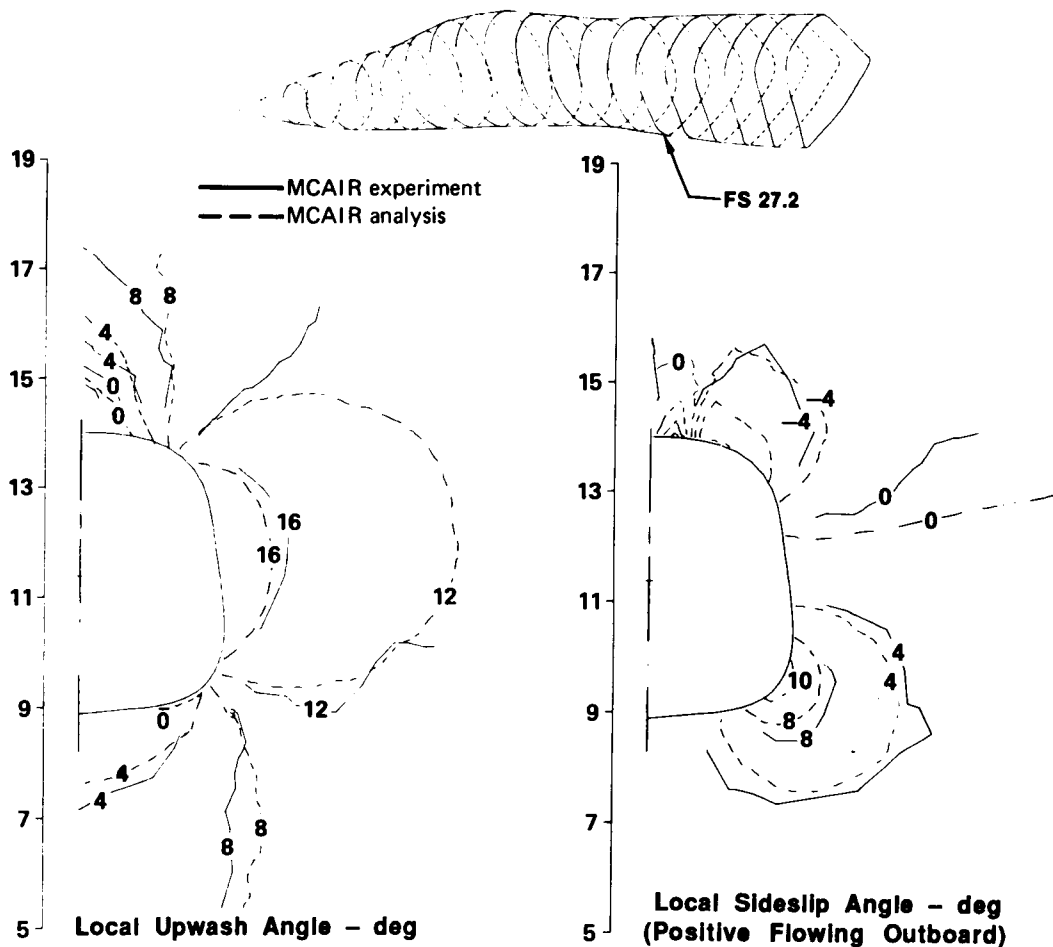

Figure 1. Flow Angularity
Forebody Flowfield
Mach 0.9   α = 10°   FS 27.2

Similar results for the Project Tailor-Mate A-1 forebody (Ref. 9) are presented in Figure 2. For this case, Mach 0.9 at high angle of attack, the level of agreement in angularity is 1° to 2°, satisfactory for most inlet integration studies.
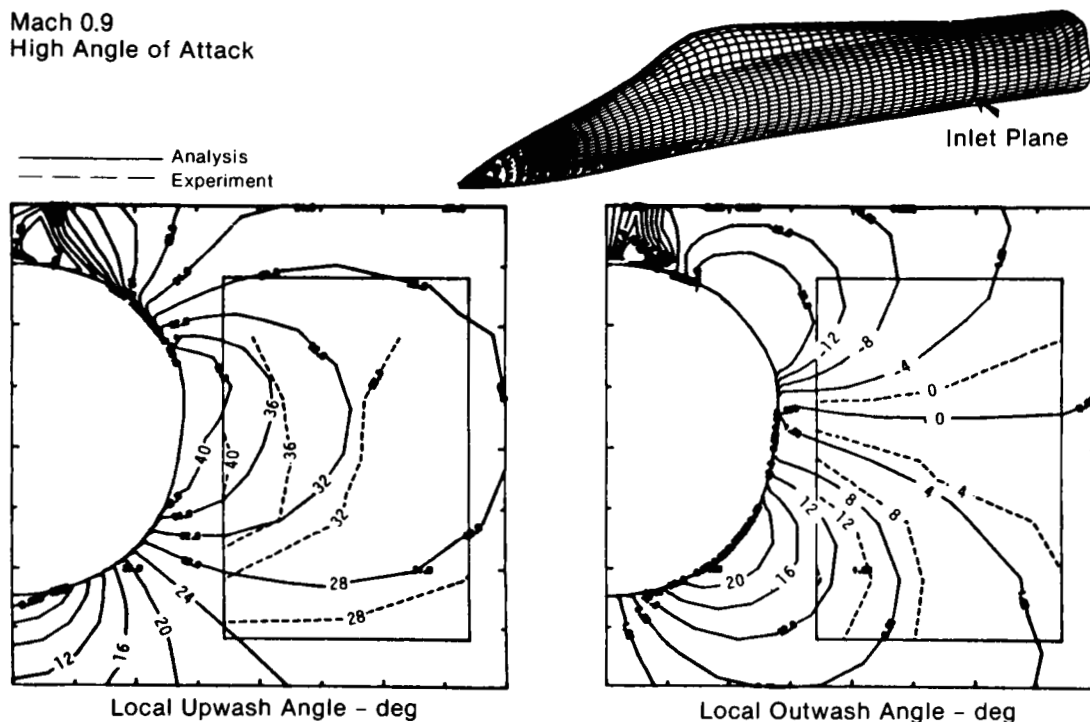
Mach 0.9
High Angle of Attack

———————— Analysis
— — — — Experiment

Inlet Plane

Local Upwash Angle – deg

Local Outwash Angle – deg

Figure 2. Flow Angularity Comparison
Tailor-Mate A-1 Forebody

Specialized post-processing capability can greatly enhance the value of CFD analysis. One example is shown in Figure 3, presenting data from an inviscid solution of the F-18 forebody at Mach 0.8, a=0°, 7.5° yaw angle (β). For this condition, some distortion was observed in the lee-side engine face flowfield. We defined the streamtube captured by the inlet on the lee side of the yawed forebody, using a predecessor of the program TRACE, described earlier.

These results showed that the captured streamtube is initially speared by the nosetip. The streamtube falls off to the lower surface of the fuselage, then sweeps around to the lee side as it moves aft. During this process, air from the boundary layer is entrained in the streamtube, leading to flow distortion in the inlet. These results were subsequently validated by a wind tunnel study using smoke to visualize this streamtube.

Program NASTD has been used to analyze vortex flowfields over various highly swept wing-body configurations. One such configuration was taken from the Cooperative Propulsion Integration Program (Reference 10). This is a joint MCAIR/USAF/NASA study. Representative results are presented in Figure 4. At Mach 0.9, $\alpha=18°$, the CFD analysis has predicted the vortex location to good accuracy. However, for supersonic flow - Mach 2.0, $\alpha=12°$ - the predicted vortex is too high, and too far inboard. Investigations are in progress to improve this type of prediction.
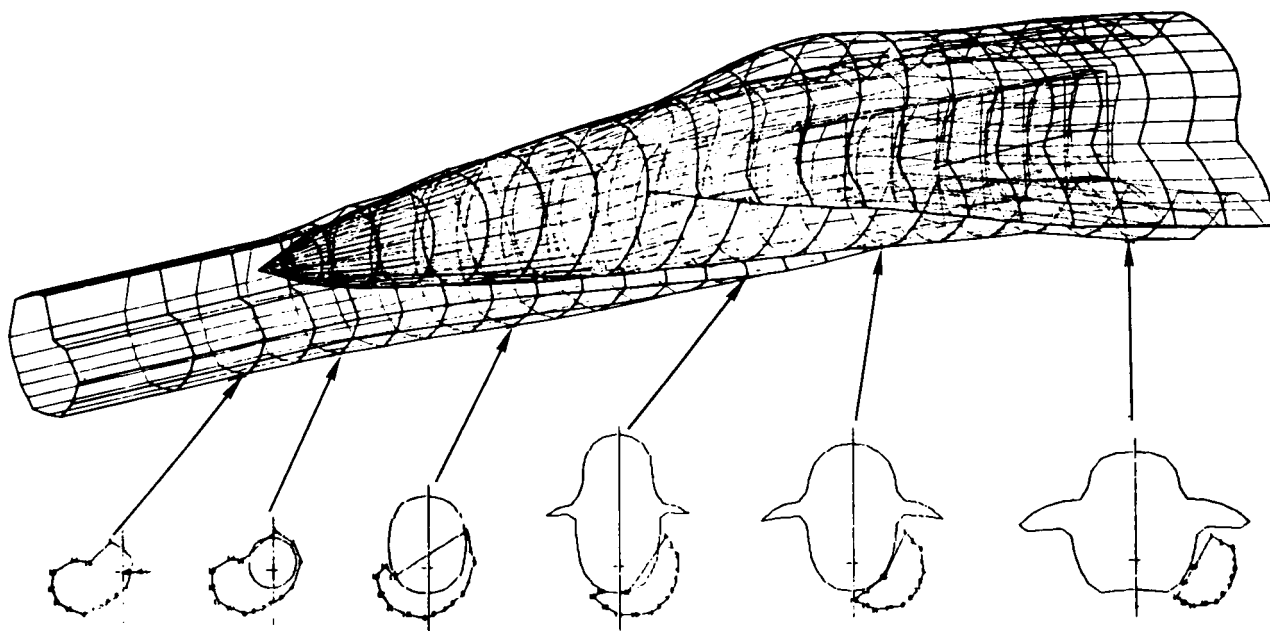
Figure 3. Forebody Flowfield Capture Streamtube Tracing
Forebody/Strake Configuration
$M_\infty = 0.8$ $\alpha = 0°$ $\beta = 7.5°$
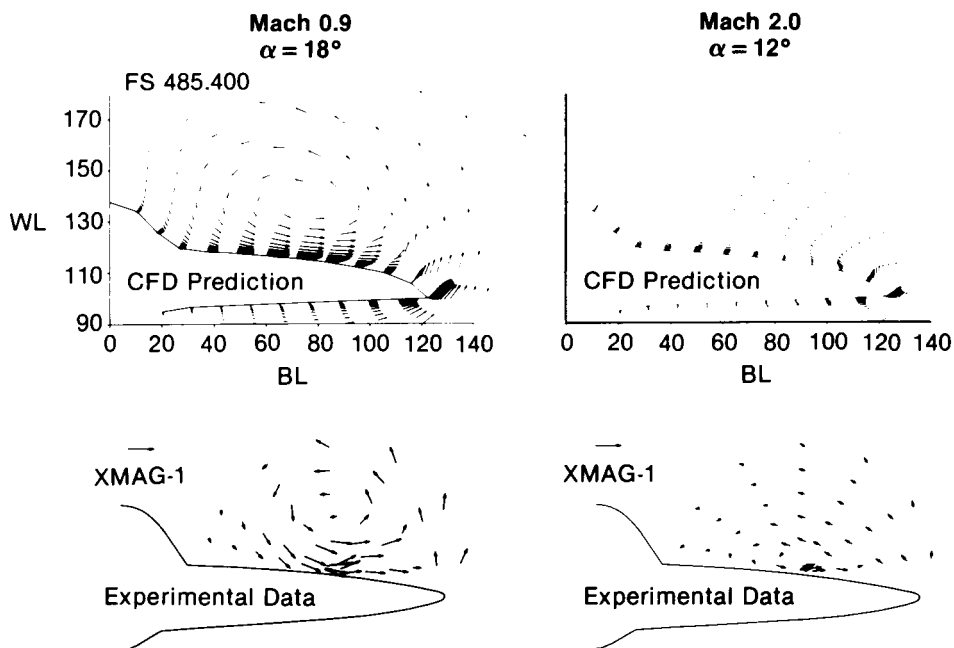

Figure 4. CFD Comparison With Test Data
Wing-Body Vortex Flowfield
Program NASTD

Isolated Inlet - Our primary tool for analysis of isolated inlets is Program FANSI. This 2D code has high geometric flexibility and a short run time for parametric investigations. Details of this method have been presented in References 1 and 2. This code has been validated for recovery predictions in 2-D inlets.

Initial validation of the FANSI code focused on total pressure recovery predictions. Sample results for several different mass flow rates are presented in Figure 5. The effect of different flow rates on shock locations is readily depicted. In the lower left portion of the figure, the dependence of recovery on mass flow is presented. These data include CFD analysis, test data, and simple estimates based on oblique shock theory. In general, the computed recovery agrees well with test data, although the accuracy falls off somewhat at the lower mass flow rates. This is attributed to spill over the sidewalls, which is not modeled in the 2D analysis.
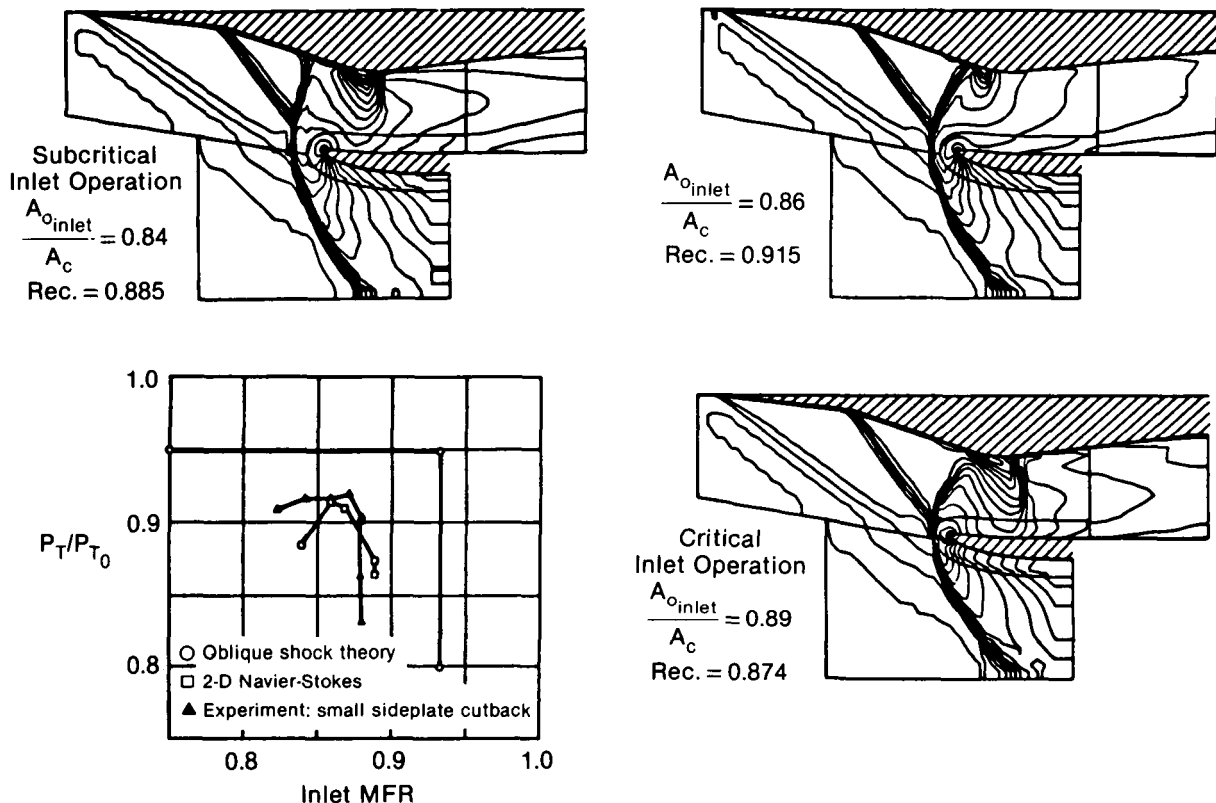

Subcritical
Inlet Operation
$\dfrac{A_{o_{inlet}}}{A_c} = 0.84$
Rec. = 0.885

$\dfrac{A_{o_{inlet}}}{A_c} = 0.86$
Rec. = 0.915

Critical
Inlet Operation
$\dfrac{A_{o_{inlet}}}{A_c} = 0.89$
Rec. = 0.874

P_T/P_{T_0}

○ Oblique shock theory
□ 2-D Navier-Stokes
▲ Experiment: small sideplate cutback

Inlet MFR

Figure 5. Effect of Mass Flow Variation on Supersonic Inlet Flowfield

With these tools, CFD results also can be used to determine the amount of bleed required to control boundary layer behavior. The results shown in Figure 6 describe the impact of different ramp bleed rates on boundary layer displacement thickness. For this example, a ramp bleed of 1% removes about two-thirds of the boundary layer, while 2% bleed completely eliminates the viscous layer.
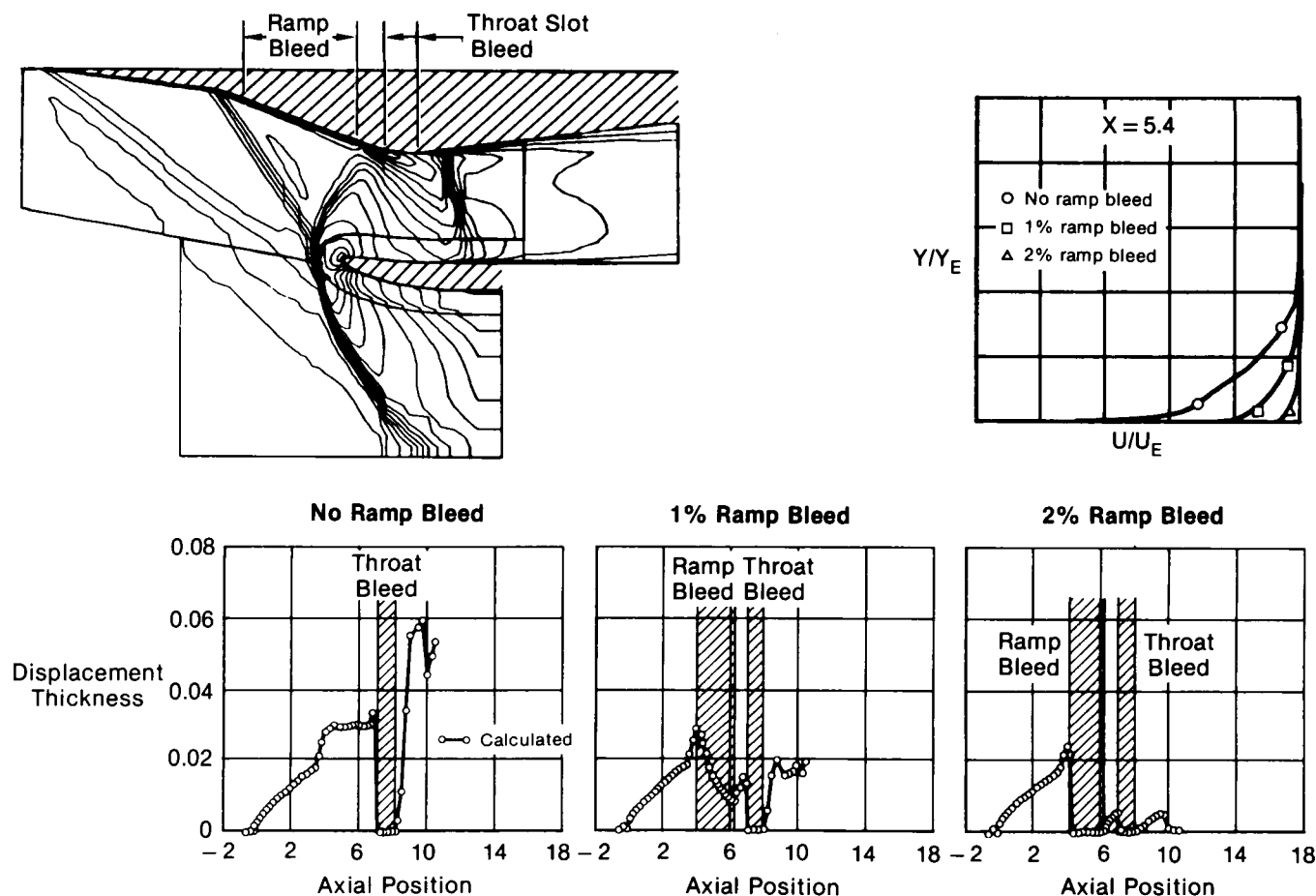
419

Figure 6. Effect of Inlet Bleed on Boundary Layer Thickness

Another key issue in inlet design is the selection of the proper lip contour. The FANSI code can be used with a C-grid about the inlet lip for accurate results in lip contour evaluation. Sample results are shown for a supersonic inlet in Figure 7. This inlet was designed for low supersonic drag and consequently features a sharp lip.

An analysis of this same configuration was done at takeoff conditions, where sharp lips often create flow quality problems. The stagnation point for the captured flow is on the lower surface of the cowl. The flow runs forward along the outside surface of the cowl, then attempts to turn sharply at the lip. The flow cannot follow this sharp contour, and separates from the inside of the lip. This separation extends downstream for about two duct heights. The analysis code also has captured a secondary vortex under the large separated zone.

The FANSI code was used to examine alternate lip designs. One concept is an actuated, drooped lip. Such a design can be used for improving low speed, high mass flow performance as well as for high a flight at higher speeds. This concept was tested recently at NASA Lewis Research Center (Ref. 11). The CFD results agree with test data in showing that a 20° lip droop provides high recovery for this inlet at high angle of attack, as shown in Figure 8.
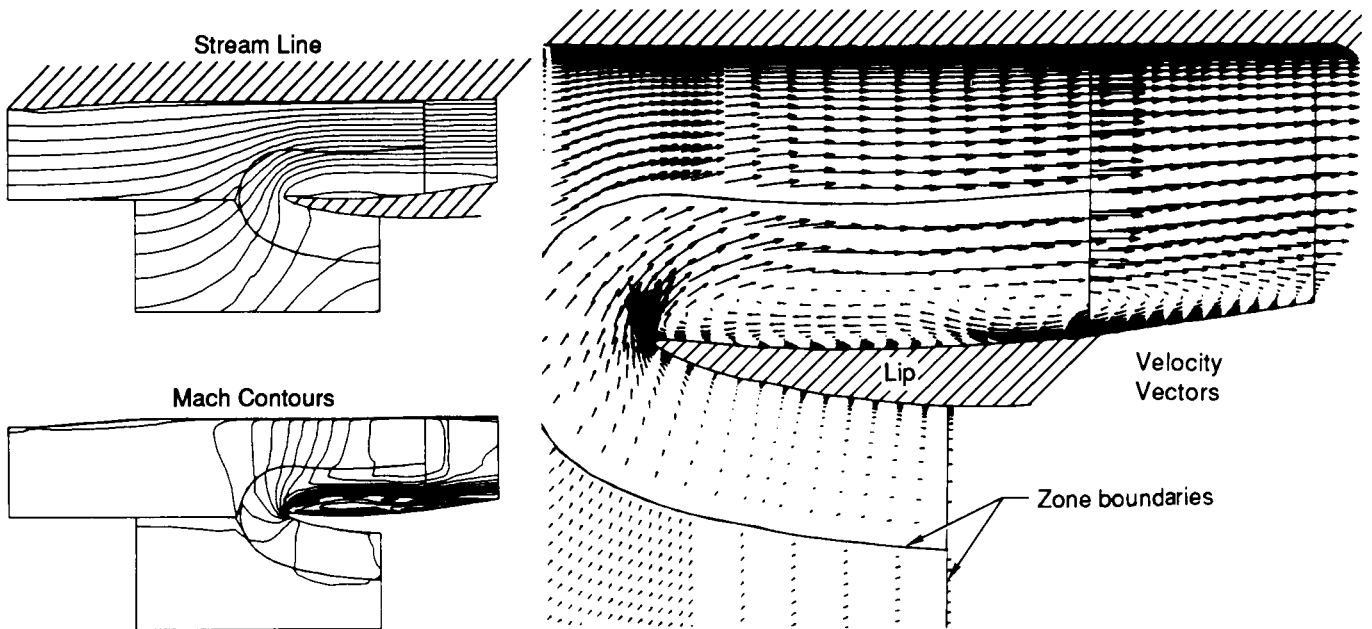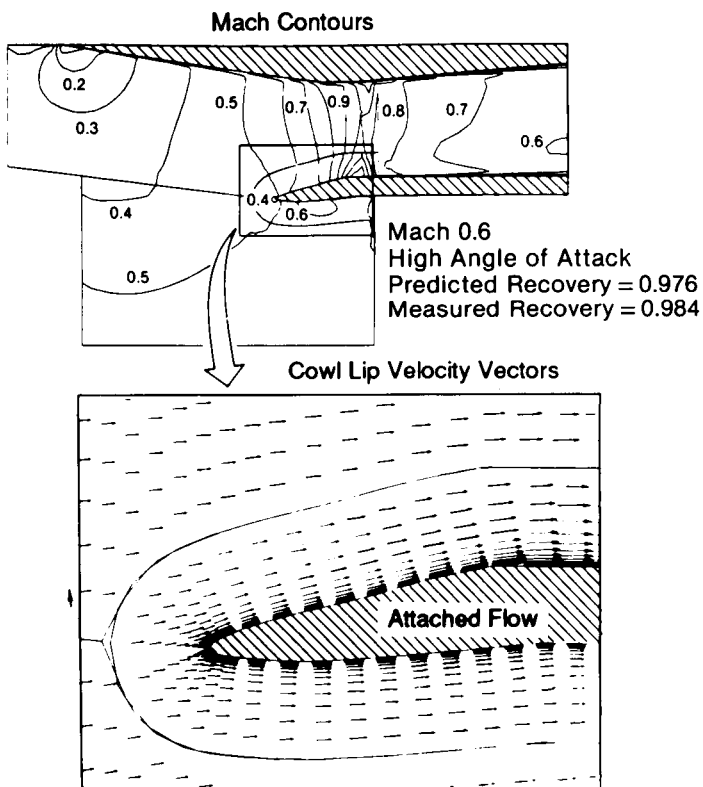
420

Stream Line

Mach Contours

Velocity Vectors

Lip

Zone boundaries

Figure 7. Cowl Lip Flow at Takeoff

$$\text{Mach } 0.1 \quad \frac{A_o}{A_c}\bigg|_{\text{inlet}} = 4.5 \quad \text{No Bleed}$$

20° Rotated Cowl Lip Configuration

Mach Contours

0.2
0.3
0.5
0.7
0.9
0.8
0.7
0.6
0.4
0.4
0.6
0.5

Mach 0.6
High Angle of Attack
Predicted Recovery = 0.976
Measured Recovery = 0.984

Cowl Lip Velocity Vectors

Attached Flow

Comparison of 2-D Flowfield Analysis Code Predictions With Experimental Total Pressure Recoveries

Total Pressure Recovery P_{t_2}/P_{t_0}

Inlet Angle-of-Attack

Cowl Rotation Schedule
$\theta = \alpha$ for $\alpha < 20°$
$\theta = 20°$ for $\alpha \geqslant 20°$

θ

Mach 0.6
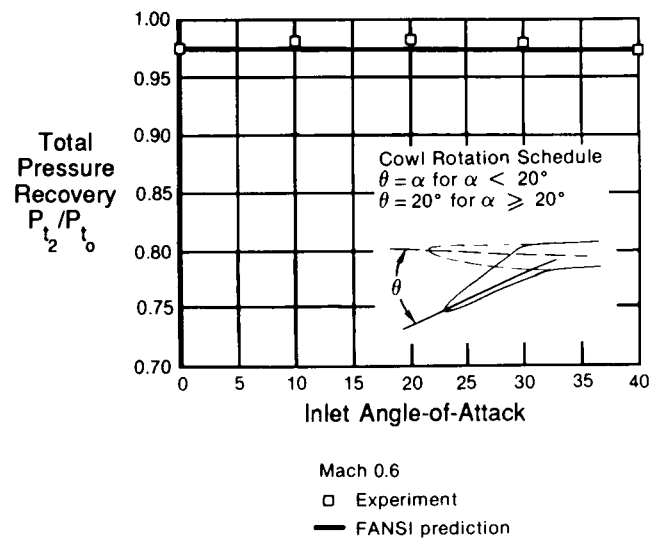□ Experiment
— FANSI prediction

Figure 8. Rotated Cowl Performance
Prediction vs Experiment

The flowfield about an undrooped cowl lip at Mach 0.6, high α is presented in Figure 9. The flow separation inside the lip is clearly revealed. The predicted pressures agree very well with experimental data. With the lip drooped 20° the flow remains attached, as shown in Figure 8. The predicted inlet recovery is nearly constant over a range of α in agreement with test data. A comparison of lip surface pressures, CFD versus test data, is presented in Figure 9. The predictions show acceptable accuracy for engineering studies.

Flowfield Predictions for Unrotated Cowl Lip Configuration at High α

Mach 0.6
Predicted Recovery = 0.931
Measured Recovery = 0.950

Mach Contours

Cowl Lip Velocity Contour

Comparison of Predicted and Experimental Cowl Lip Static Pressures
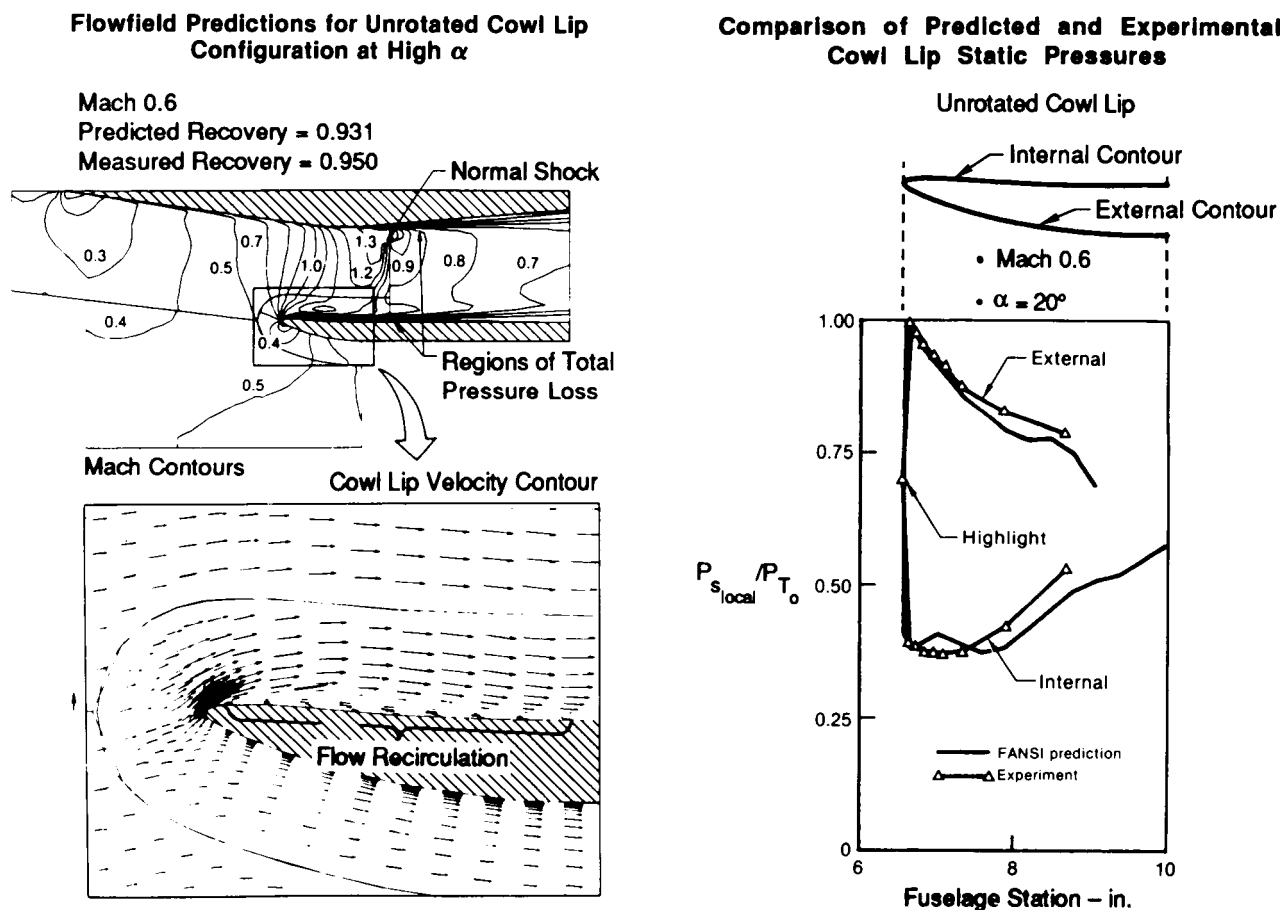
Unrotated Cowl Lip

Figure 9. CFD Predictions Compare Favorably With Experimental Results

The 3D codes X3D and NASTD have been used to predict flow over 3D isolated inlets. An inviscid solution from X3D is presented in Figure 10. This example presents flow at Mach 2.0 through the A-1 inlet from the Tailor-Mate program. This test was conducted by General Dynamics, under contract to the Air Force. An inviscid NASTD solution is presented in Figure 11, for an inlet tested by MCAIR at NASA Lewis Research Center (Ref. 11). Both these isolated inlet solutions used a two-zone mesh from ZGRID. The boundary between zones extends forward from the inlet highlight, and can be seen in these two figures. The oblique and normal shocks pass cleanly through the zone boundary, and the spill over the inlet lip and sidewalls is captured.
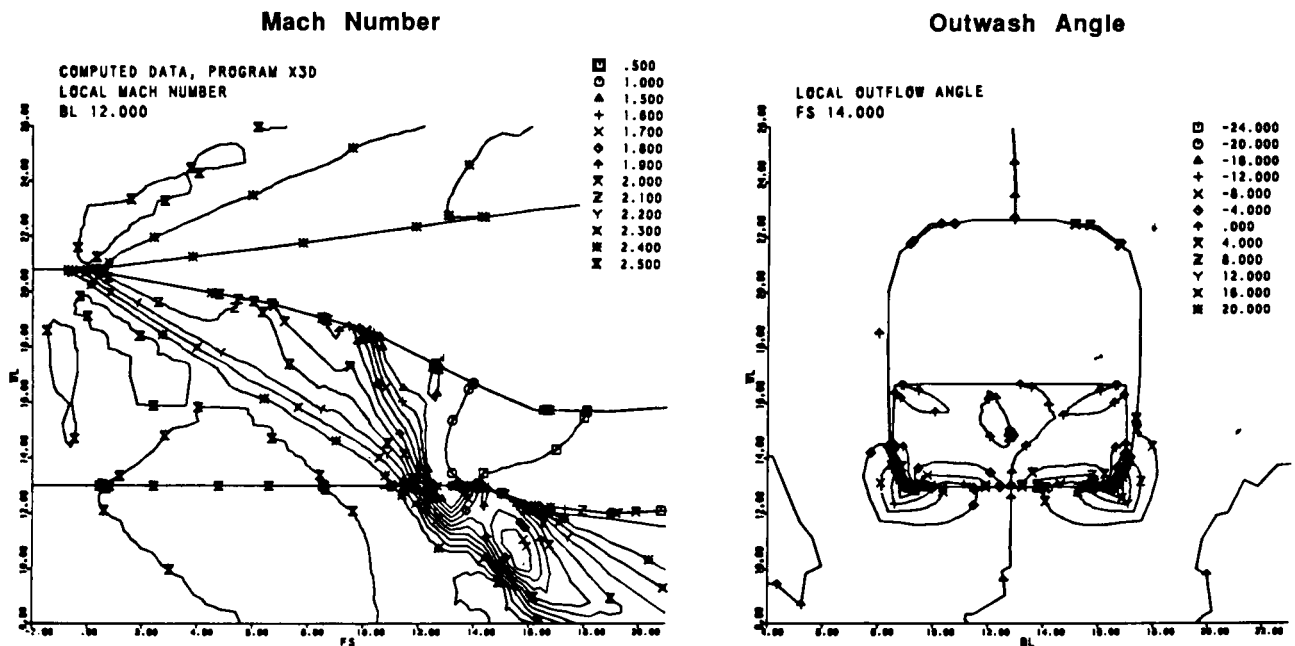
422

Mach Number **Outwash Angle**

Figure 10. Tailor-Mate A-1 Inlet
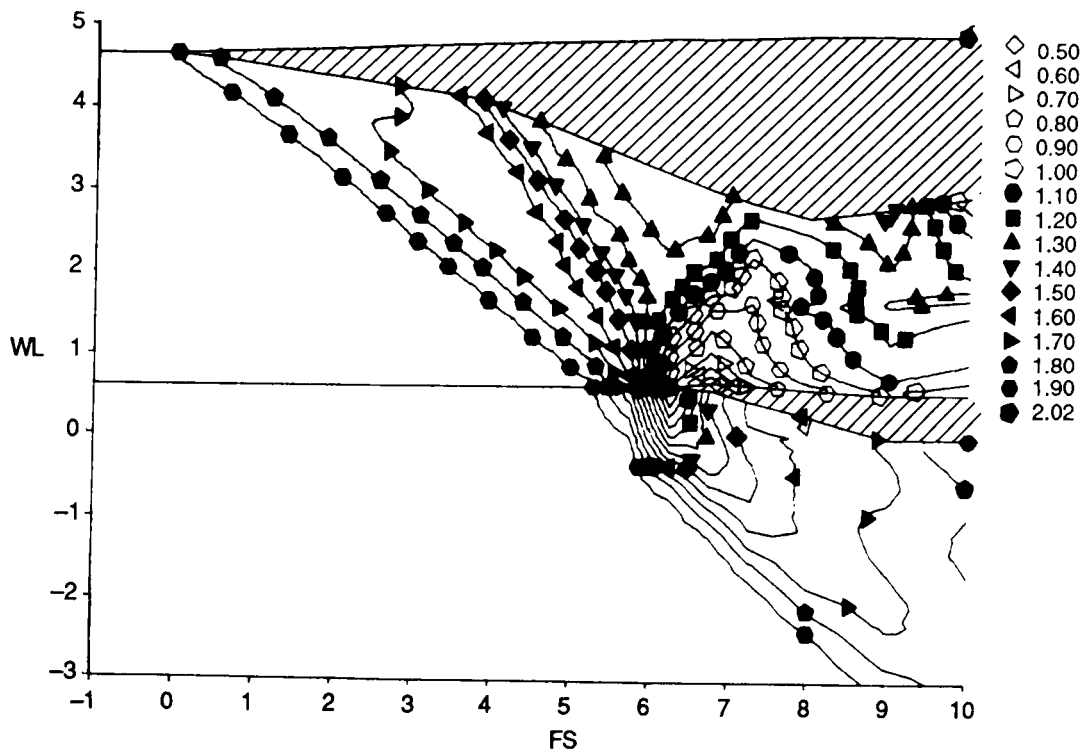Mach 2.5, With Bleed
X3D Inviscid Solution

Figure 11. Supercruise Inlet Analysis, Mach 2.0
Program NASTD

Isolated diffuser - Another key element of the propulsion system is the inlet diffuser. Detailed analysis of diffuser flowfields can be a critical concern in aircraft design. Sample analysis results are presented for several cases.

As part of the Ref. 12 study, MCAIR designed and tested several concepts for compact, highly offset diffusers. Many of these concepts were analyzed using Program X3D. One result, for the so-called B19 diffuser, is presented in Figure 12. Strong viscous interactions are present as a result of a high rate of diffusion and a high offset. The boundary layer is completely separated from the upper wall for about half the diffuser length, as seen in the total pressure contour plots. The predicted area-averaged total pressure recovery is in reasonable agreement with the test data, as seen in Figure 13.

Velocity Vectors

Total Pressure Recovery
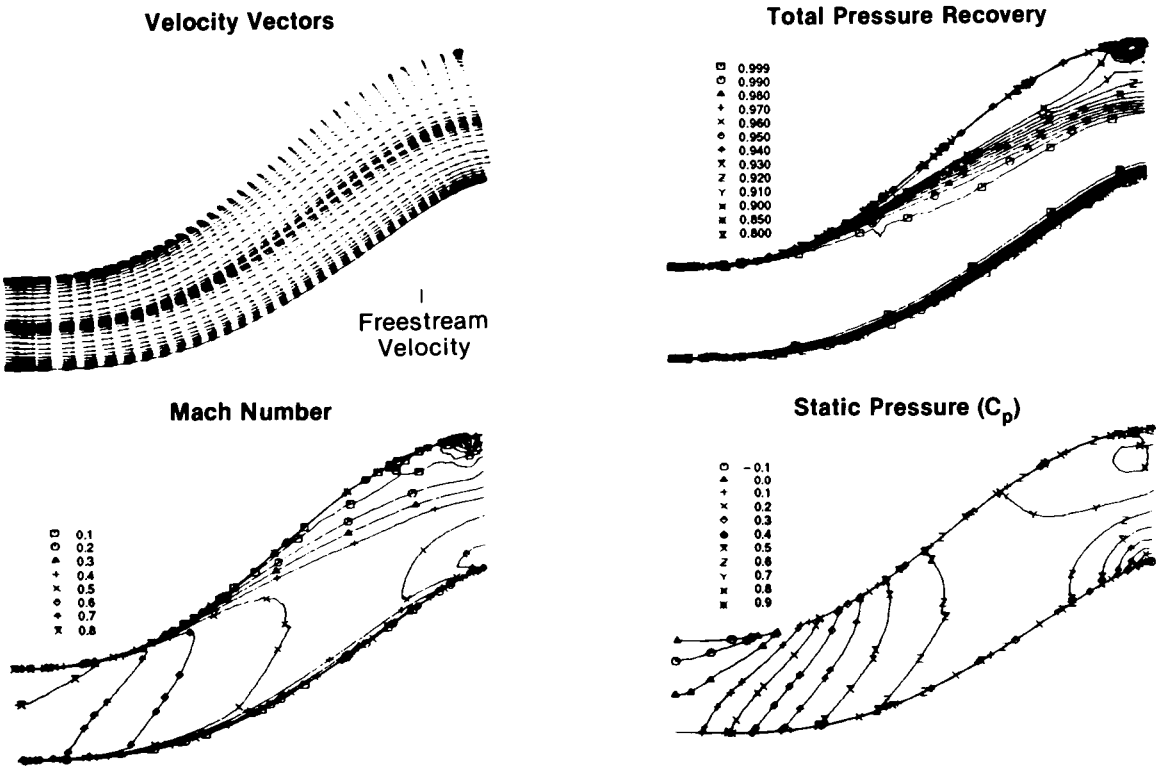
Mach Number

Static Pressure (C$_p$)

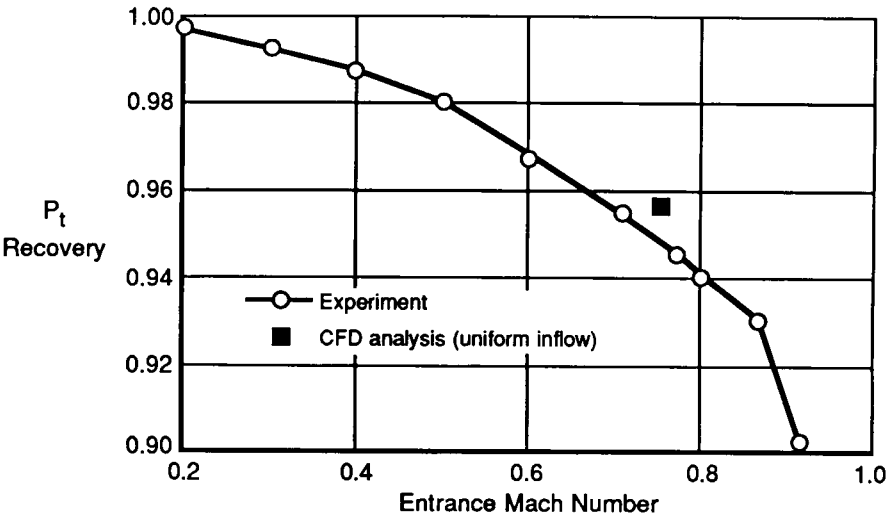Figure 12. B19 Diffuser Flowfield
Uniform Inflow   Mach 0.777

Figure 13. Recovery Prediction
B19 Diffuser

In the same study, the X3D code was used to assess the impact of diffuser entrance conditions on exit properties. Predictions were made for cases with uniform inflow, and with measured inflow data taken from entrance flowfield surveys. The predicted total pressure profiles for these two cases are presented in Figure 14. These results show that the exit total pressure distribution is significantly affected, even though the core entrance Mach number is practically the same (0.005 difference).
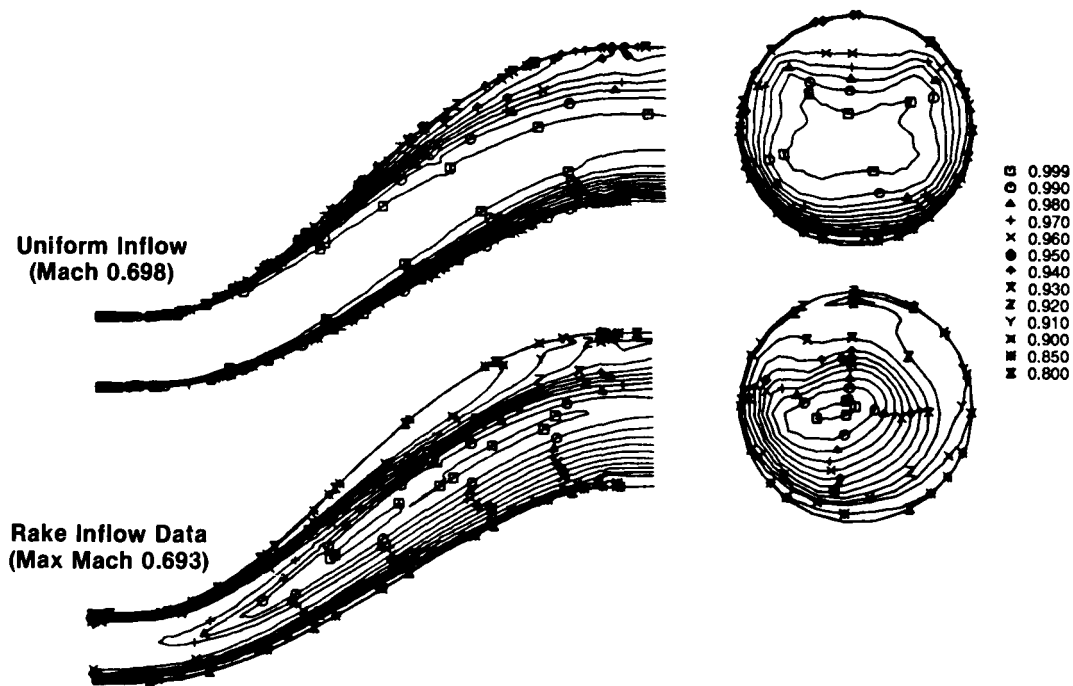
Uniform Inflow
(Mach 0.698)

Rake Inflow Data
(Max Mach 0.693)

□	0.999
◎	0.990
▲	0.980
✦	0.970
✕	0.960
●	0.950
✦	0.940
✕	0.930
✖	0.920
Y	0.910
✕	0.900
✖	0.850
✕	0.800

Figure 14. Effect of Inflow Condition
ADII Diffuser
Total Pressure Recovery
CFD Analysis

The diffuser exit area-averaged recovery for these two inflow conditions is presented in Figure 15. The recovery predictions with uniform inflow are unacceptably high compared with test data. By recognizing the proper inflow, but otherwise performing the same analysis, the recovery prediction error is reduced considerably.

Integrated Forebody-Inlet - The X3D code is used extensively for analysis of integrated forebody-inlet combinations. Some results have been presented in Ref. 13. The initial application was for a representative fighter geometry. Predictions were validated by comparison with wind tunnel data for Mach 0.8, a=0°. These results are presented in Figure 16. The comparison is made for pressures on the inside of the lower inlet lip. Good accuracy is demonstrated at both a flight idle mass flow rate (116 lb/sec) and near-maximum mass flow (155 lb/sec).
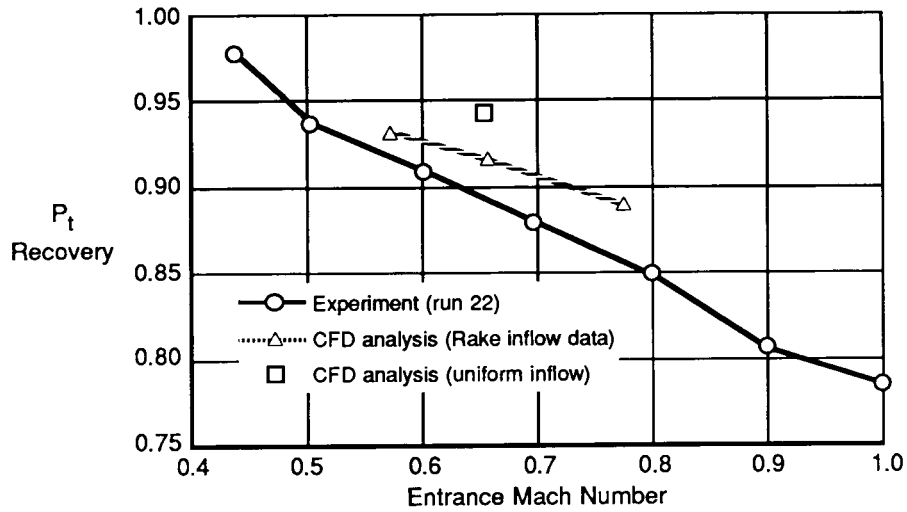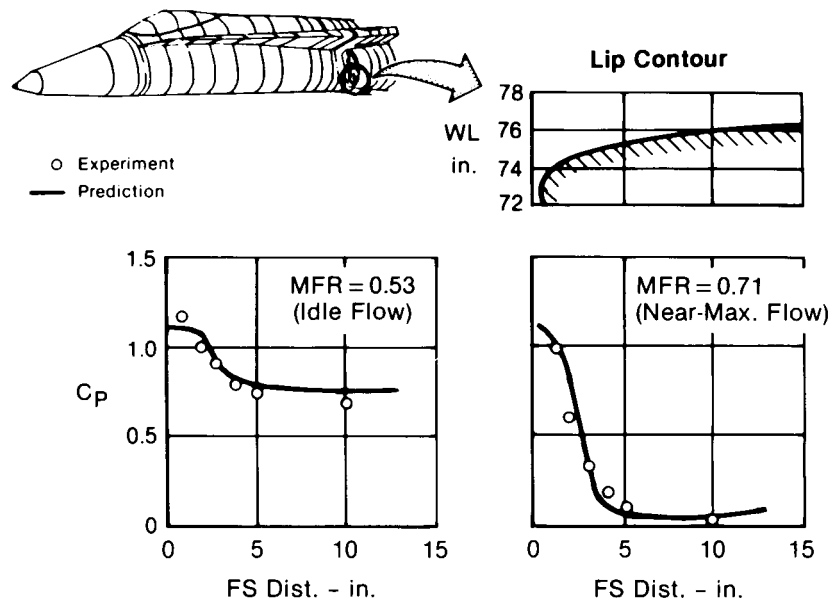
425

Figure 15. Recovery Prediction
ADII Diffuser

Figure 16. Inlet Internal Pressure
Bottom Lip Static Pressure   Mach 0.8 $\alpha = 0°$

The analysis code also provides valuable information which was not acquired in the wind tunnel program. In one example, CFD was used to investigate the effect of engine airflow rate on forebody pressures, as shown in Figure 17. Predicted data are shown for the two mass flow rates presented previously and also for zero net mass flow through the inlet. The integrated mass flow is zero, but the method allows local flow into or out of various portions of the inlet entrance plane. These results show a forebody pressure impact which extends upstream more than 100 inches from the inlet entrance.
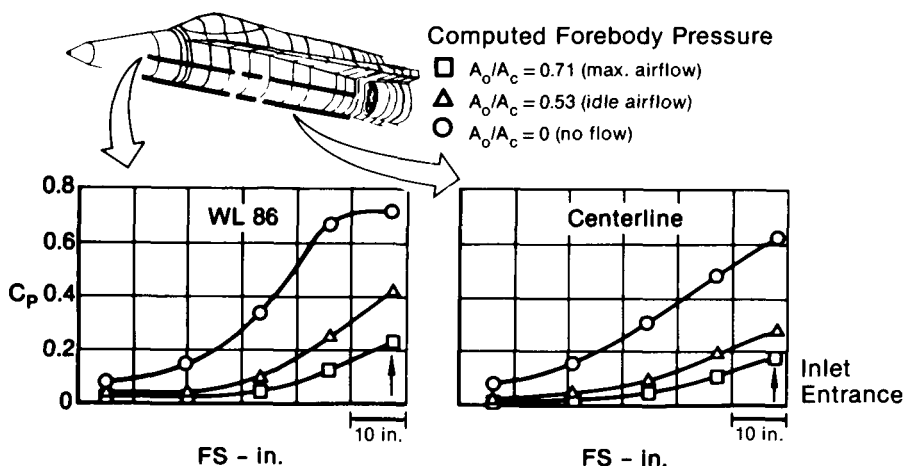

Figure 17. Effect of Engine Airflow on Fighter Forebody Pressure
Mach 0.8 $\alpha = 0°$

Similar analysis has been conducted for other inlet ducts. As seen in Figure 18, the analytical data agree well with test data for the inside of the lower lip at Mach 0.67, a=10°. The extensive data provided by CFD analysis also allowed detailed examination of additional flow properties, such as surface static pressure (right side of Figure 18).

The development of the total pressure field in an inlet duct is presented in Figure 19. At Fuselage Station (FS) 225, which is about five inches inside the inlet, the viscous layer is very thin and behaves as a simple boundary layer. In the middle of the duct (FS 245), the viscous layer is much thicker and is not behaving as a simple boundary layer on the inboard wall: the total pressure contours are not parallel to the wall. Three inches in front of the engine face (FS 260), a complex pattern of total pressure loss is predicted. A more detailed examination of this and other solutions was used by engineers to suggest duct modifications which were incorporated into production AV-8B's.

Nozzle Internal Flow - All three zonal Navier-Stokes codes (FANSI, NASTD, X3D) have been applied to nozzle internal flow. The objectives have been to predict effective vector angles, nozzle internal losses, wall heating distributions, and the overall development of flow properties in the duct. Sample results are presented in Figure 20 for the throat region of a 3D nozzle analyzed using NASTD.
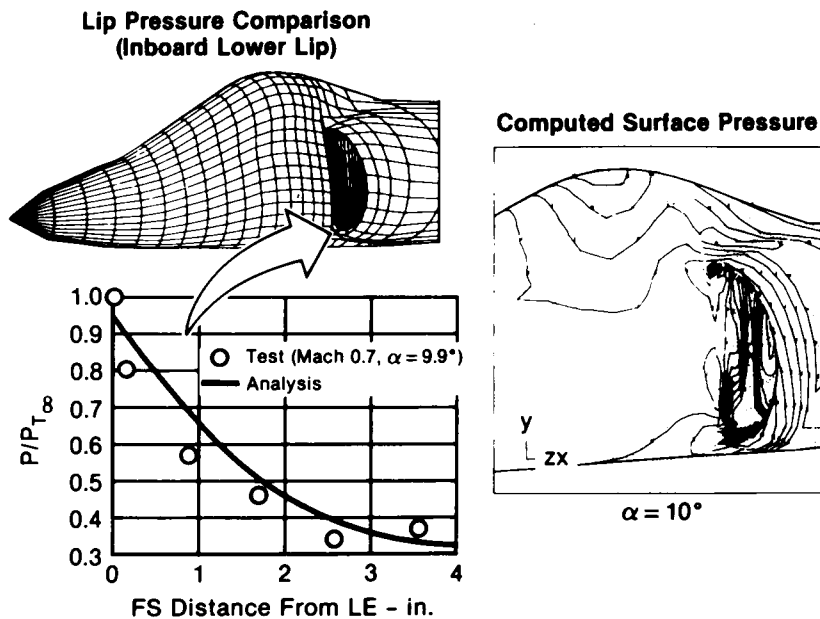
427

**Lip Pressure Comparison
(Inboard Lower Lip)**

Computed Surface Pressure

O Test (Mach 0.7, $\alpha = 9.9°$)
— Analysis

P/P_{T_∞}

FS Distance From LE – in.

$\alpha = 10°$

Figure 18.   Forebody-Inlet Viscous Analysis
Mach 0.67 $\alpha = 10°$ Cruise Airflow

**FS 225
(Inlet Entrance)**

**FS 245
(Mid-Duct)**

**FS 260
(Engine Face)**

P_t/P_{t_∞}

☐	0.999
☻	0.960
▲	0.920
+	0.880
✕	0.840
◆	0.800
✦	0.760
✗	0.720
Z	0.680
Y	0.640
✖	0.600
✳	0.560
✗	0.520
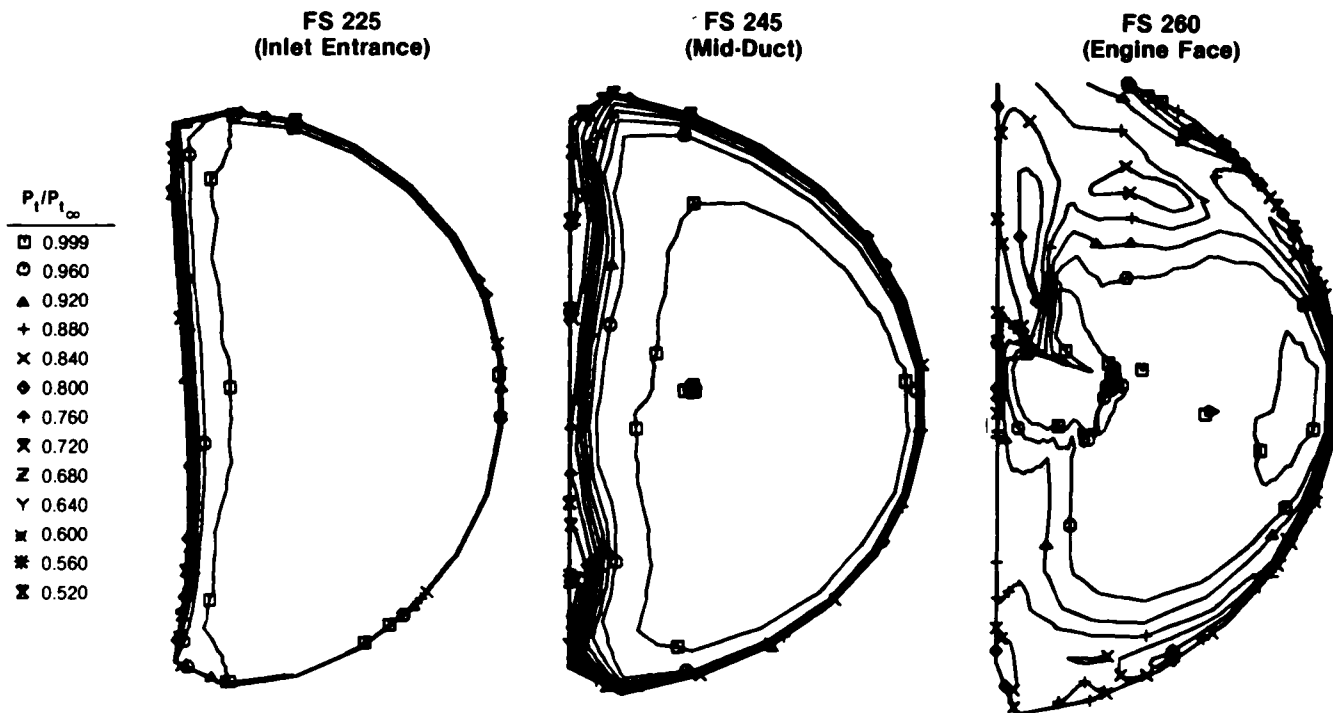
Figure 19.   Computed Total Pressure Distribution
Fighter Diffuser   Mach 0.67 α 0.67   Cruise Airflow

428

12.00

11.81

11.22

10.83

10.44

WL

10.08

9.87

9.28

8.80

8.60

O	0.400	O	1.800
□	0.500	◐	1.900
△	0.600	●	2.000
▽	0.700	■	2.200
◇	0.800	▲	2.400
◁	1.000	▼	2.600
▷	1.200		
○	1.600		

69.39   70.17   70.94   71.72   72.50   73.28   74.06

59.00   69.78   70.56   71.33   72.11   72.89   73.67
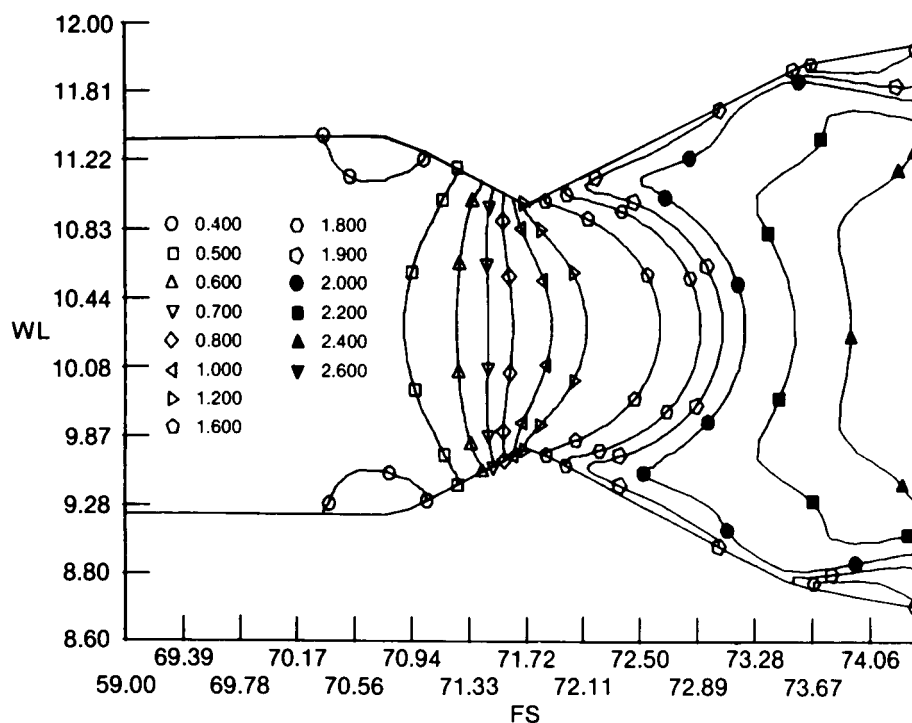
FS

Figure 20. Mach Contours for 3D Nozzle Internal Flow

Nozzle External Flow - The initial development of the velocity-splitting method was aimed at analysis of nozzle-afterbody configurations at transonic speeds. Several applications have been published in past years (Refs. 4 and 5). Most of our validation efforts for afterbody-nozzle external flow have been based on the test data from the Advanced Nozzle Concepts (ANC) Program, Ref. 14.

A comparison between test and analysis for the baseline axisymmetric nozzle from the ANC program is presented in Figure 21. This case is for a dry power (non-afterburning) nozzle setting at Mach 0.9, α=0°. At the time this analysis was performed (1983), we could not represent the vertical tail. Omission of this component did affect the prediction accuracy somewhat over the upper surface behind the vertical tail, but the agreement was good elsewhere. The most important figure of merit for afterbody flowfield prediction is the drag accuracy. For this case, our analysis agreed with test data, with a one count (0.0001) error in drag coefficient.

Similar results for Mach 2.0 are presented in Figure 22. Again, excellent agreement is obtained for the baseline axisymmetric nozzle - one count error in pressure drag. For the 2D nozzle, the pressure drag error is not as good - seven counts.
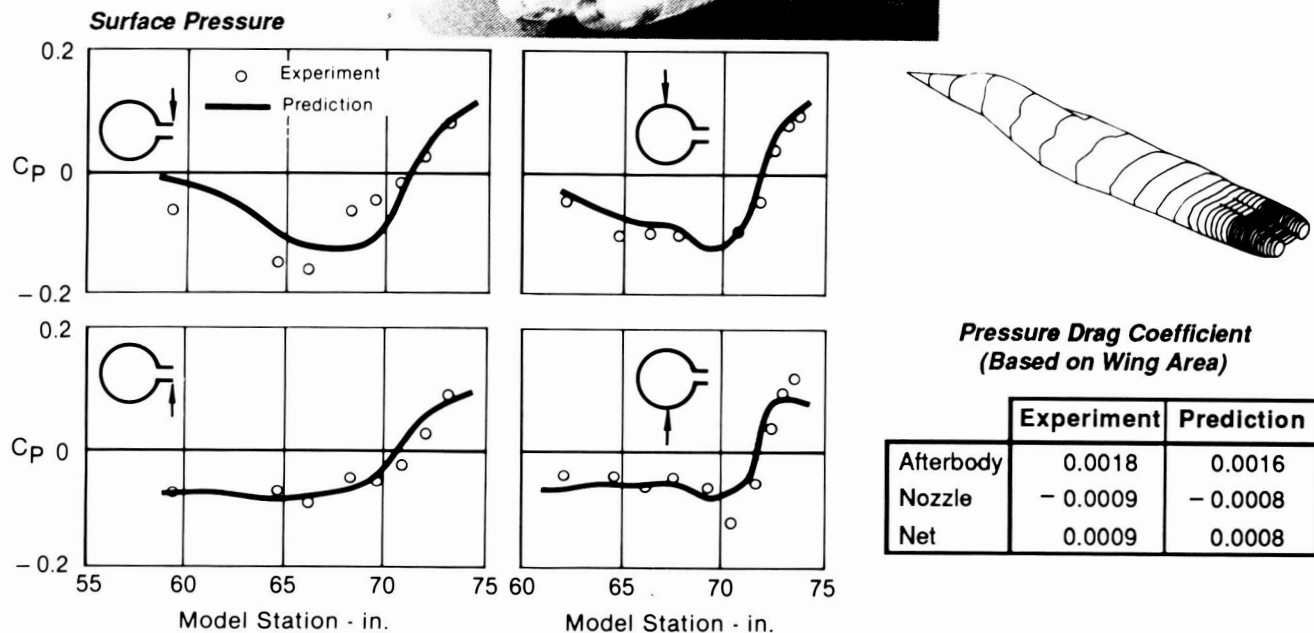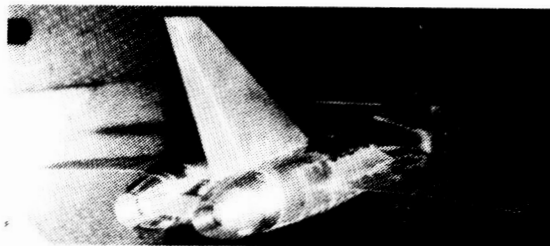
429

Figure 21. Pressure and Drag Prediction
ANC Air-to-Surface Model
Baseline Axi Nozzle
Dry Power   Mach 0.9 $\alpha = 0°$

More recently, NASTD has been used to analyze external flow over nozzles tested in the USAF/MCAIR program "High Performance Supercruise Nozzles" (Contract F33615-84-C-3003). Comparisons between CFD and test data at Mach 0.9 are presented for the Pratt and Whitney Tandem Disk 2D C-D Nozzle in Figure 23. These comparisons present the longitudinal variation of surface pressure on the upper surface nozzle centerline, and also the boundary layer profile at the start of the nozzle. These data show generally good agreement in the surface pressure and excellent agreement in the approach boundary layer profile. The afterbody pressure drag is predicted to about 10% accuracy; the CFD drag coefficient based on wing area is 0.0056; the coefficient computed from measured data is 0.0050.

Our prediction accuracy for 3D afterbody drag is not consistent however. It is excellent for some cases, disappointing for others. Current usage therefore is generally restricted to examination of flowfield features and relative variations in drag due to moldline changes.
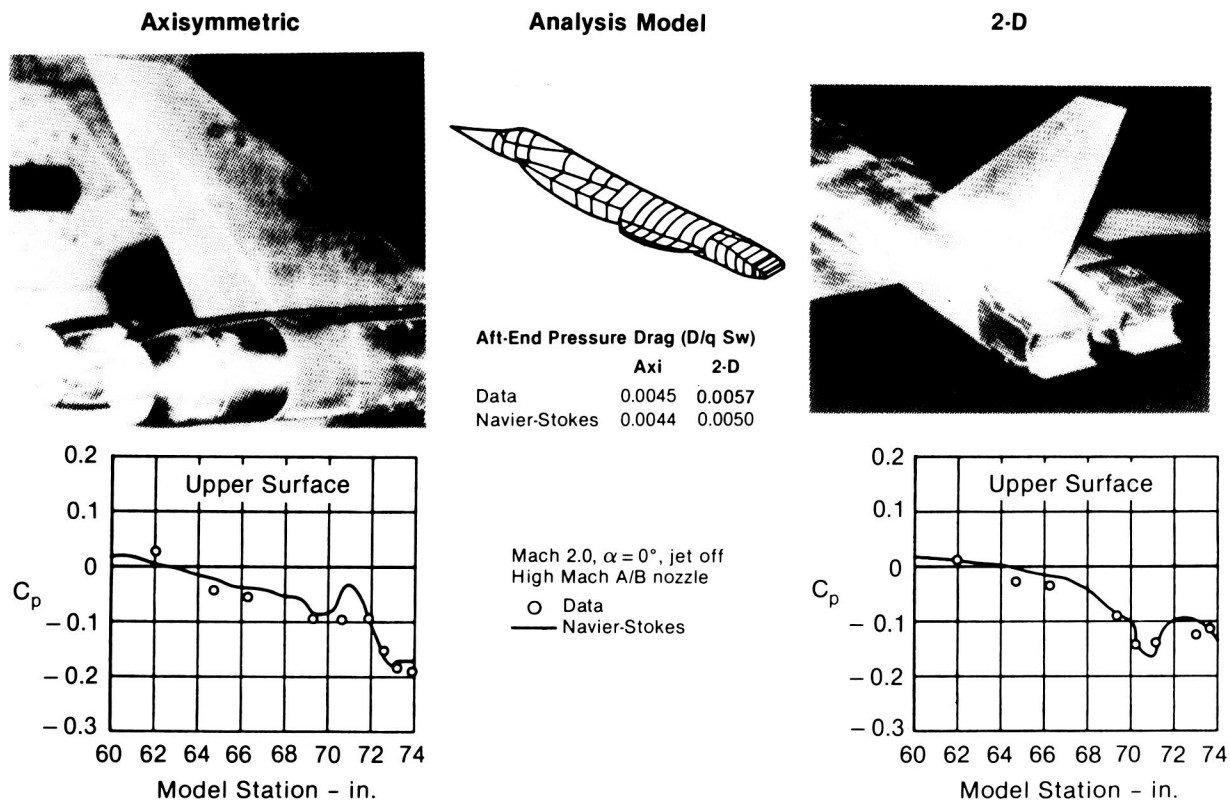
Axisymmetric   **Analysis Model**   **2-D**

Aft-End Pressure Drag (D/q Sw)

	Axi	2-D
Data	0.0045	0.0057
Navier-Stokes	0.0044	0.0050

Mach 2.0, $\alpha = 0°$, jet off
High Mach A/B nozzle
○   Data
——   Navier-Stokes

Upper Surface — left plot: C_p vs Model Station – in.

Upper Surface — right plot: C_p vs Model Station – in.

Figure 22. Code Verification for Supercruise Nozzle Integration

Surface Pressure Comparison   **Boundary Layer Profile**

Pressure Coefficient (CP) vs Model Station – in.

MS 62.0   68.8   74.4

○   Experimental data from supercruise nozzle test
——   Navier-Stokes prediction (program NASTD)

Afterbody Drag
Measured 0.0050
Predicted 0.0056

Boundary Layer Rake

Distance From Surface in.

○   Experimental data from supercruise nozzle test
——   Navier - Stokes prediction (program NASTD)

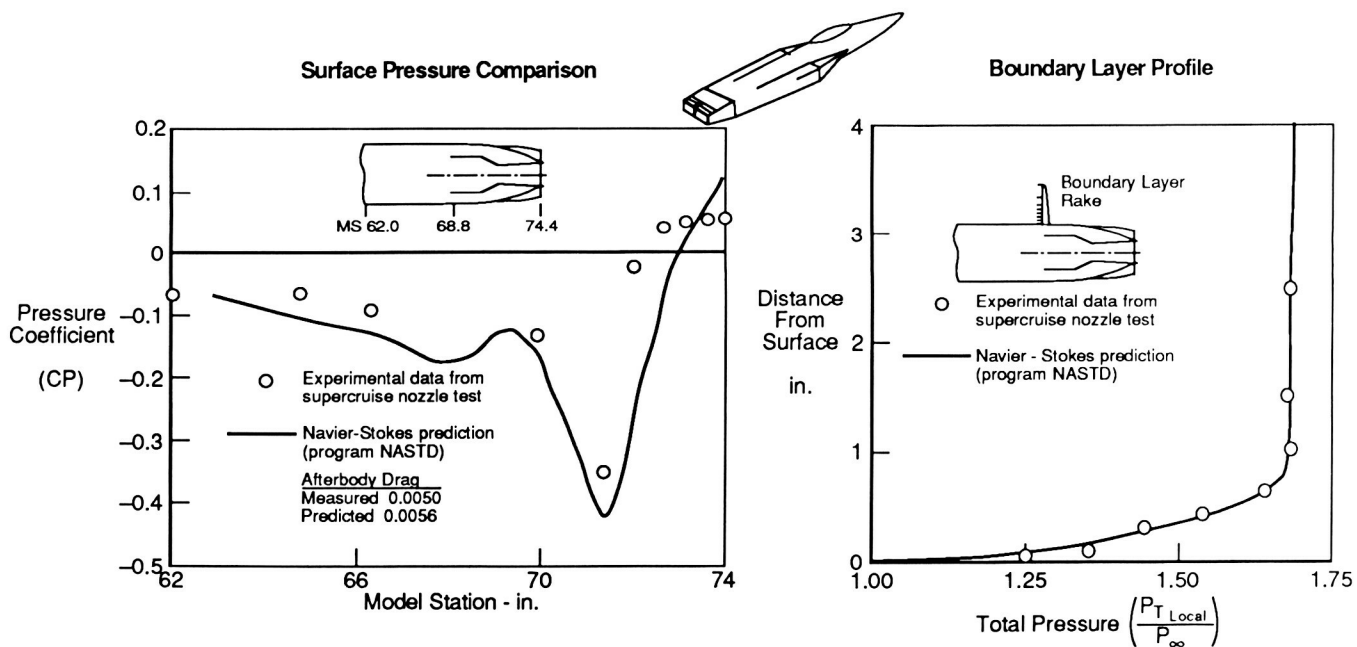Total Pressure $\left(\dfrac{P_{T_{Local}}}{P_\infty}\right)$

Figure 23. Aftbody NASTD Calculation - Tandem 2-D C-D Nozzle, Mach 0.9

431

Combined External/Internal Nozzle Flow - The FANSI program has been modified to analyze 2D nozzle flow for a range of nozzle configurations including Single Expansion Ramp Nozzle (SERN) designs and ejector nozzles. As with other applications, this effort is based on using multiple, non-overlapping computational zones.

The boundary conditions in the internal and external flowfields can be set independently. Inflow data can be uniform or arbitrarily specified. Sample results for transonic analysis of a hypersonic nozzle design are presented in Figure 24.
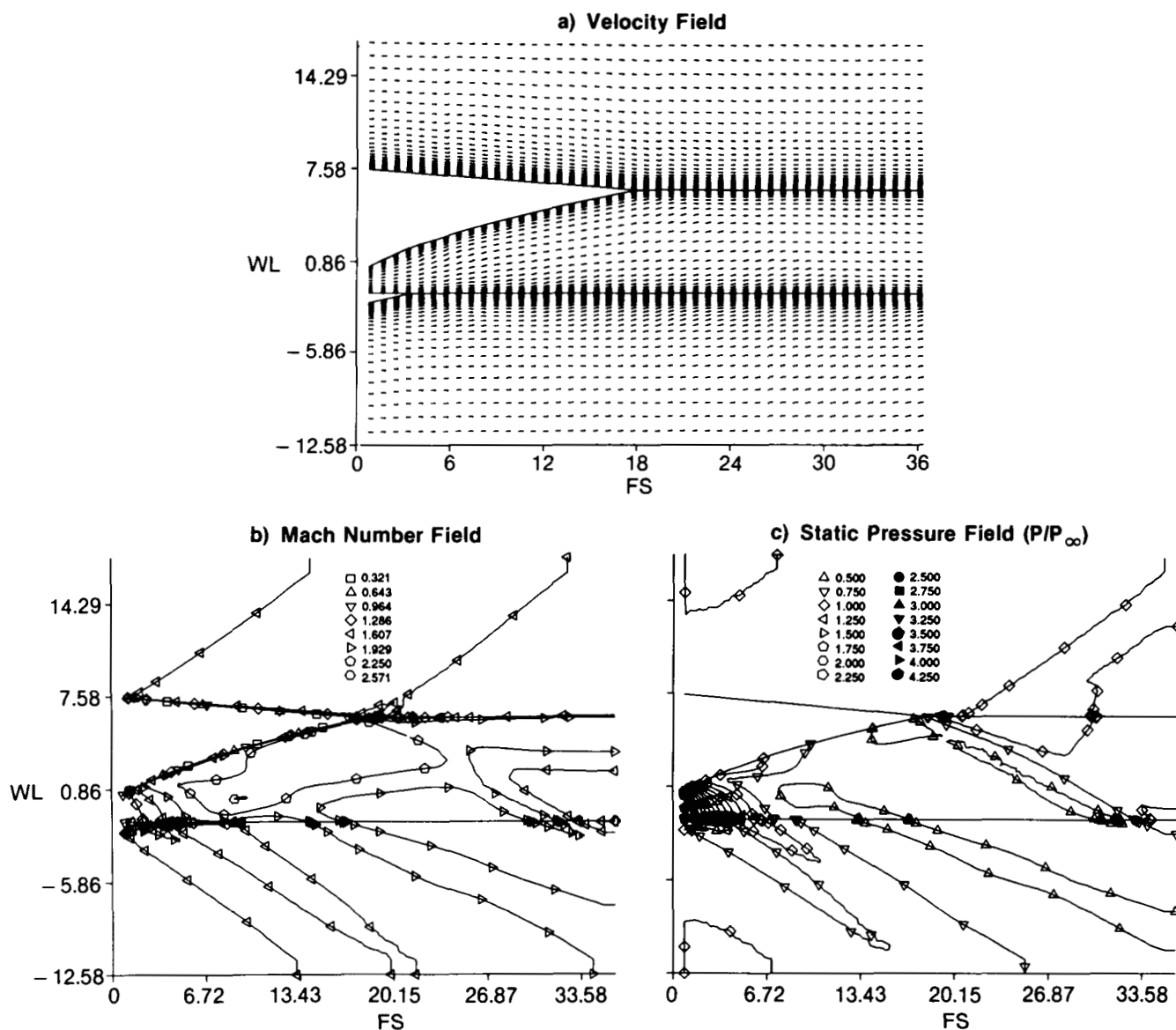
a) Velocity Field

b) Mach Number Field

c) Static Pressure Field (P/P$_\infty$)

Figure 24. SERN Flowfield Analysis, Program FANSI

$M_\infty = 1.5$ $P_{t_j}/P_\infty = 8.07$ $T_{t_\infty} = 727°R$ $T_{t_j} = 560°R$

Recent emphasis has been on the analysis of ejector nozzles. Sample FANSI results for a nozzle with one secondary stream are presented in Figure 25.
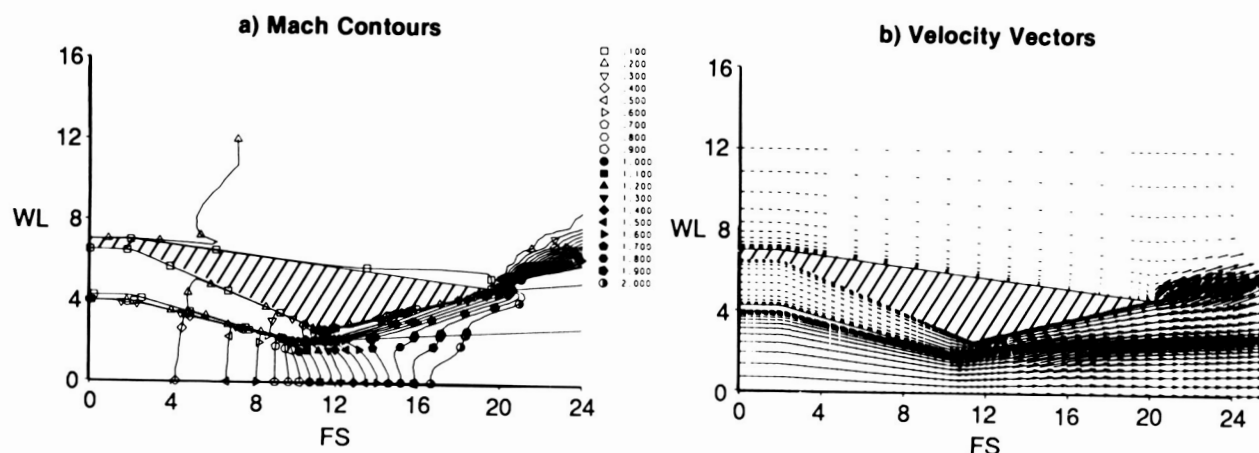
a) Mach Contours

b) Velocity Vectors

Figure 25. Ejector Nozzle Flowfield, Program FANSI
$M_\infty = 0.2$

SUMMARY

Computational Fluid Dynamics procedures are becoming familiar tools in the design of aircraft propulsion components and integrated systems. To an increasing degree, CFD methods are entering the design process in several ways:

- Evaluating and screening alternate concepts.

- Refining design concepts.

- Improving our understanding of complex flowfields.

- Selecting wind tunnel parametric variations.

- Designing wind tunnel instrumentation.

- Interpreting test data.

Current CFD methods can be used very effectively, but the engineering labor and computer costs of CFD application are often very high. More efficient algorithms always will be desired. Improvements are needed in developing trusted algorithms which do not require tuning to specific problems. Current solution costs, which have improved greatly over the years, are still often seen by project engineers as excessive.

At MCAIR, the problem in CFD applications generally is in the mesh generation tools, rather than the flow solvers. Mesh generation methods often are based on limited geometry input schemes which have been developed for specific classes of configurations. The trend is toward man-in-the-loop, interactive grid generation. This offers maximum geometric capability and is an approach which is being actively pursued at MCAIR. However, this approach carries an operational cost: it usually takes a skilled CFD engineer at the workstation providing the interactive guidance.

A complete CFD system will, in our opinion, require interactive grid generation capability for new, complex problems. However, one goal is to provide non-interactive mesh generation methods for as many classes of problems as possible. All examples presented in this paper used non-interactive mesh generation.

Perhaps the most striking advances recently have been in graphical display tools (hardware and software) for CFD results. These tools have gone a long way toward convincing engineering managers that CFD is an indispensable tool.

The current challenge at MCAIR is to integrate CFD into the engineering environment. To accomplish this, we need to:

- Streamline the handling of data.

- Develop a base of validated CFD experience.

- Modify "research" grid and flowfield codes into "engineering" codes which do not require a CFD expert for most applications.

- Provide the support tools to allow the project engineer to accomplish his goals quickly.

- Educate our personnel in the potential and the limitations of CFD analysis.

The last item is perhaps the most significant. In the past, extravagant claims have been made for CFD analysis ("electronic wind tunnel"). The challenge now is to bring this technology into the enginneering workplace on a routine basis. This will be accomplished by making realistic claims and then delivering the promised data on time.

CFD has become an accepted tool in many areas of aerospace engineering. It has the potential to change drastically the way we do business. But we have only scratched the surface in exploiting the current technology.

The major growth in engineering applications will result from the CFD community recognizing the needs of project engineers and managers. Project personnel need tools which can be applied quickly, with confidence, to realistic flight vehicle geometries. Validation data should be available to establish confidence in the quality of results by comparison with wind tunnel or flight data. Guidelines are required to set all input parameters which are not defined by the physical problem to be analyzed. The codes must be usable without iterative adjustment of input data to obtain the needed results. The high potential of CFD will be realized when this technology is used effectively by project engineers, in conjunction with an array of other tools, to design a vehicle which will accomplish a specific mission.

434

ACKNOWLEDGMENT

REFERENCES

1. Bush, R.H., "External Compression Inlet Predictions Using an Implicit, Upwind, Multiple Zone Approach," AIAA Paper 85-1521-CP, presented at AIAA 7th Computational Fluid Dynamics Conference, Cincinnati, 15-17 July 1985.

2. Bush, R.H., P.G. Vogel, W.P. Norby, and B.A. Haeffele, "Two Dimensional Numerical Analysis for Inlets at Subsonic through Hypersonic Speeds," AIAA Paper 87-1751, presented at AIAA/SAE/ASME 23rd Joint Propulsion Conference, 29 June - 2 July 1987, San Diego.

3. Cosner, R.R., "Fast Navier-Stokes Solution of Transonic Flowfield about Axisymmetric Afterbodies," AIAA Paper 80-0193, presented at AIAA 18th Aerospace Sciences Meeting, Pasadena, 14-16 January 1980.

4. Cosner, R.R., "Relaxation Solution for Viscous Transonic Flow about Fighter-Type Forebodies and Afterbodies," AIAA Paper 82-0252, presented at AIAA 20th Aerospace Sciences Meeting, Orlando, 11-14 January 1982.

5. Mace, J.L., and R.R. Cosner, "Analysis of Viscous Transonic Flow Over Aircraft Forebodies and Afterbodies," AIAA Paper 83-1366, presented at AIAA/SAE/ASME 19th Joint Propulsion Conference, Seattle, 27-29 June 1983.

6. Yaros, S.F., and R.R. Cosner, "Evaluation of a Velocity-Split Solution of the Navier-Stokes Equations for Fighter Forebodies," AIAA Paper 84-2160, presented at AIAA 2nd Applied Aerodynamics Conference, Seattle, 21-23 August 1984.

7. Buning, P.G., and Steger, J.L., "Graphics and Flow Visualization in Computational Fluid Dynamics," AIAA Paper 85-1507, proceedings of AIAA 7th Computational Fluid Dynamics Conference, Cincinnati, 15-17 July 1985.

8. Zulauf, J.M., "Real-Time Imaging in a CFD Environment," Computer Graphics World, Vol. 10, No. 8, pg. 40 (August 1987).

9. Cawthon, J.A., E.L. Crosthwait, and P.P. Truax, "Supersonic Inlet Design and Airframe-Inlet Integration Program (Project Tailor-Mate)," AFFDL-TR-71-124 (May 1973).

10. Bare, E.A., Reubush, D.E., Haddad, R., Hathaway, R.W., and Compton, M., "Investigation of a Delta-Wing Fighter Model Flow Field at Transonic Speeds," AIAA Paper 87-1749, presented at AIAA/SAE/ASME/ASEE 23rd Joint Propulsion Conference, San Diego, 29 June - 2 July 1987.

11. Norby, W.P., B.A. Haeffele, and R.R. Burley, "Isolated Testing of Highly Maneuverable Inlet Concepts," NASA CR 179544, December 1986.

12. Lee, C.C., and W.A. Price, "Subsonic Diffusers for Highly Survivable Aircraft," AFWAL-TR-86-3025 (Final Report for the Period September 1982 - April 1986), May 1986.

13. Cosner, R.R., "Integrated Flowfield Analysis Methodology for Fighter Inlets," AIAA Paper 85-3071, presented at AIAA/AHS/ASEE Aircraft Design, Systems and Operations Meeting, 14-16 October 1985, Colorado Springs. Also presented as AIAA Paper 86-1463, presented at AIAA/SAE/ASME/ASEE 22nd Joint Propulsion Conference, 16-18 June 1986, Huntsville.

14. Hiley, P.E., and D.L. Bowers, "Advanced Nozzle Integration for Supersonic Strike Fighter Application," AIAA Paper 81-1441, presented at AIAA/SAE/ASME 17th Joint Propulsion Conference, Colorado Springs, 27-29 July 1981.