Propulsive-Jet Flow Field Analysis Using The Three-Dimensional Navier-Stokes Equations

C. L. Reed

NASA

NASA Contractor Report 177508

Propulsive-Jet Flow Field Analysis Using The Three-Dimensional Navier-Stokes Equations

C. L. Reed

GENERAL DYNAMICS CORP.
FORT WORTH DIVISION
P.O. BOX 748
FORT WORTH, TEXAS 76101

TABLE OF CONTENTS

PRECEDING PAGE BLANK NOT FILMED

PAGE II INTENTIONALLY BLANK

LIST OF FIGURES

PRECEDING PAGE BLANK NOT FILMED

INTRODUCTION

A critical aspect of advanced aircraft design is the
effect of exhausting hot, high-velocity gas into the
flow field adjacent to the aircraft. Jet-entrainment
produced when a propulsive-jet is exhausted straight back
from a fighter-type aircraft can account for up to 25% of the
total airplane drag. Exhausting the hot gas in any direction
other than straight back, such as in propulsive-lift or
thrust-reversing, can have a significant, and sometimes
devastating, impact on the aircraft performance. To date,
these effects have only been determined through extensive
wind tunnel testing. Computational models of these flows
have not been available because of the complexity of the flow
fields and because of the extensive computer resources
required. But now, with new numerical algorithms and high
capability computers, advancements can be made in the
development of viscous, 3-D analysis procedures for
propulsive-jet flow fields.

Several approaches have been taken to determine the
characteristics of high velocity jets exhausting into a
relatively slower crossflow. Adler and Baron (reference 1)
use an integral method to predict the characteristics of a
circular turbulent jet in crossflow. Baker and Orzechowski

1

(reference 2) have used a parabolized Navier-Stokes solver along with a rather complicated start-up procedure to analyze a jet in a crossflow. Numerous others have also attempted solutions of this type. These are summarized in reference 2. The current approach is to model flow fields containing propulsive-jets by solving the 3-D time-dependent Navier-Stokes equations. Although more expensive, this method should have a much wider range of applicability.

This report summarizes the work carried out to test and validate a propulsive-jet flow field analysis method which is based on the 3-D Navier-Stokes equations.

BACKGROUND

In 1983, an effort was initiated within General Dynamics to develop a propulsive-jet flow field analysis capability using the 3-D Navier-Stokes equations. Because of General Dynamic's experience in using the code for inlet analysis, the full 3-D time-dependent Navier-Stokes (N-S) code developed by J. S. Shang (reference 3) was selected as the baseline code to build on. Some very preliminary results were obtained for a cylindrical body with a sonic jet exiting normal to a subsonic freestream. Although these results indicated that much work was yet to be done, they were encouraging and allowed confidence to be placed in the N-S code's ability to model propulsive-jet problems.

In late 1983, a cooperative program was initiated with the NASA Ames Research Center to continue the 3-D N-S code work. Under this program, NASA Ames supplied both computer resources and jet-in-crossflow test data. After numerous code modifications, the NASA flat plate plume model test case (reference 4) was analyzed. A schematic of this test case is shown in Figure 1. The results were much more encouraging and indicated a well defined plume within the flow field. Figure 2 shows the results for a velocity ratio of 3.66 and a zero-gradient upstream boundary condition.
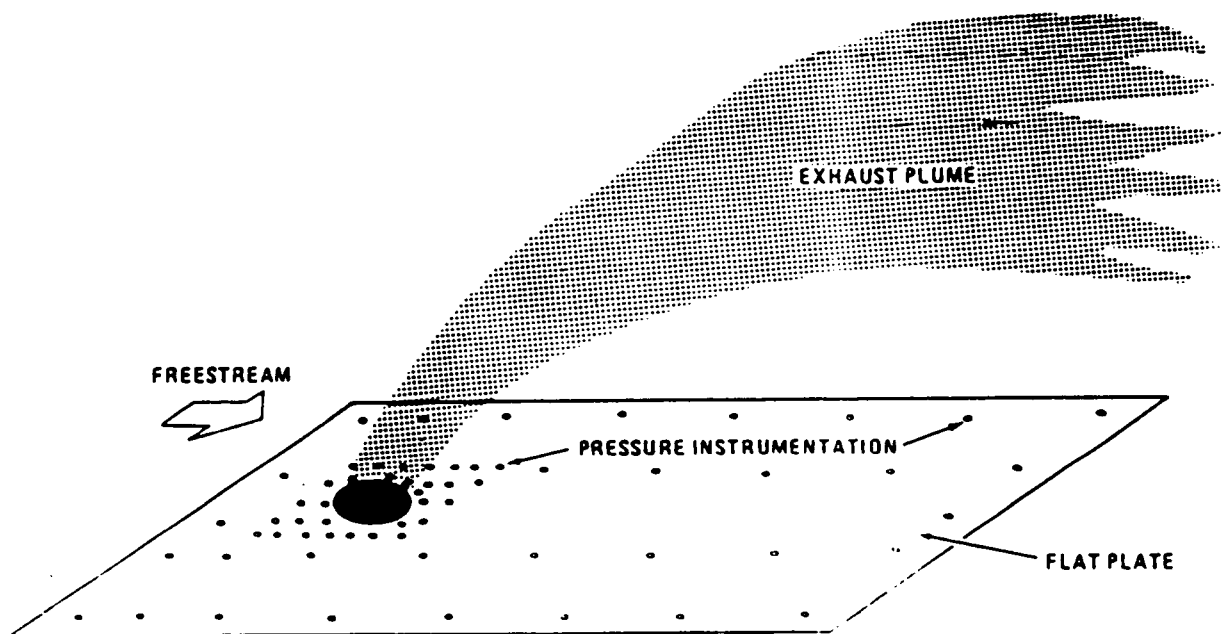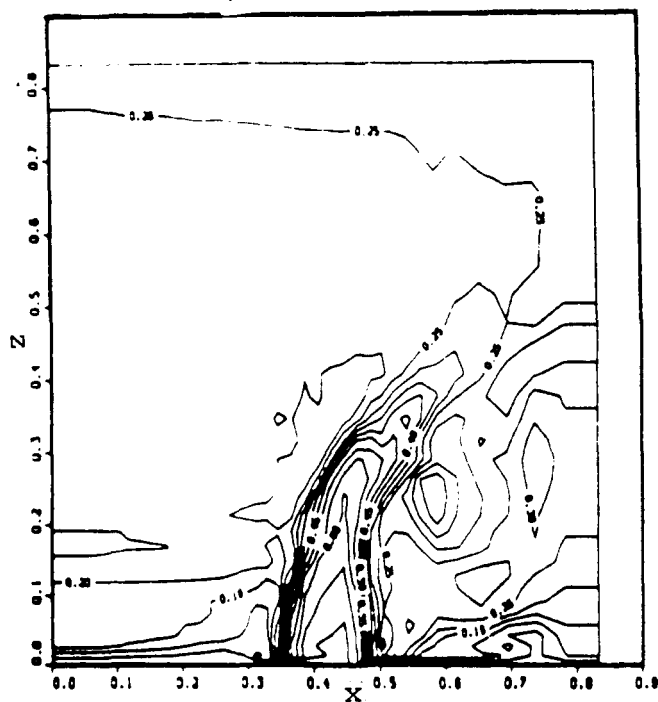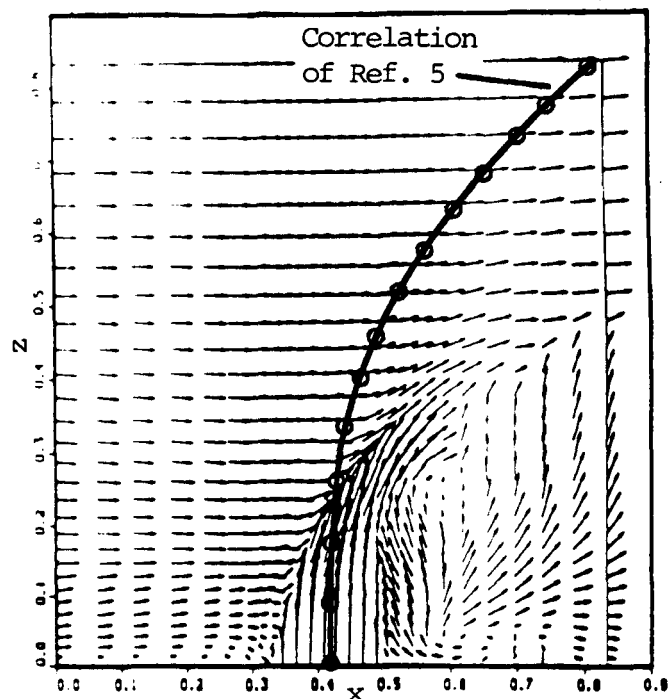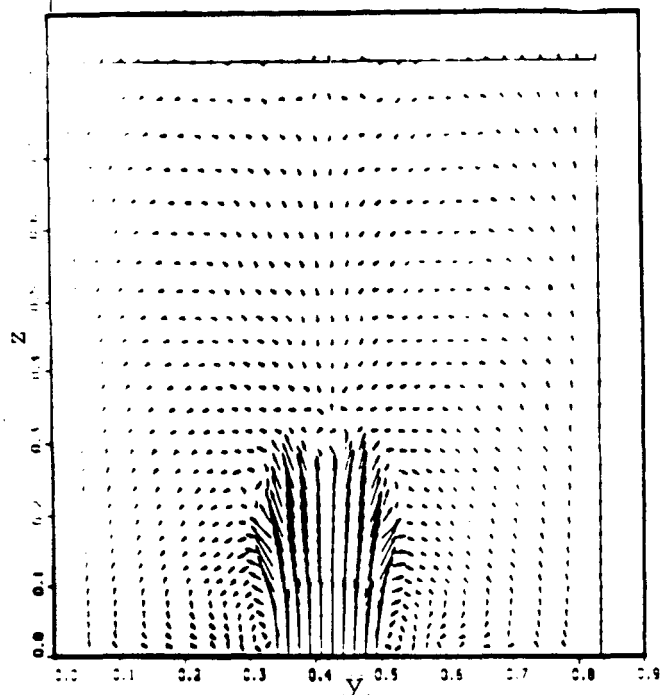
Figure 1   Schematic of the NASA Flat-Plat Plume Model

a) Side View - Mach Contours

b) Side View - Velocity Vectors

c) Front View - Velocity Vectors

d) Top View - Velocity Vectors

Figure 2   Computational Results for the NASA Flat-Plate
Plume Model, V_j/V_∞ = 3.66, Zero-Gradient
Upstream Boundary Condition

5

No test data were available for this case, however, the results were compared with the empirical jet path equation of reference 5. Although the computational results do not match the empirical jet path, they do indicate that the 3-D N-S code can model highly vectored propulsive jets. In a later test case, a problem was discovered in using a zero-gradient upstream boundary condition. As a temporary solution to this problem, the velocities were specified on the upstream boundary. This type of boundary condition is more consistent mathematically, though still not an exact modeling of the physics. Figure 3 shows the results using a specified upstream velocity and a jet-to-freestream velocity ratio of 8.0. Again, the results are compared to the empirical jet path equation of reference 5. Laser doppler velocimeter (LDV) test data were available for this case and are qualitatively compared to the computationally derived data in Figure 4. The results from these test cases are very encouraging and reinforce our confidence in the ability of the 3-D N-S code to model these complex flow regions.
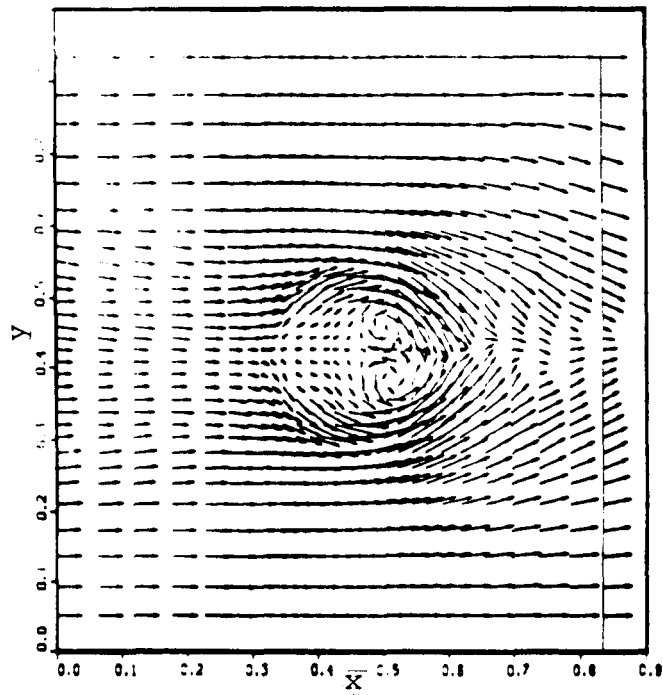
a) Side View - Mach Contours
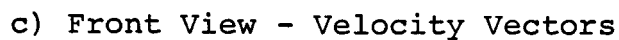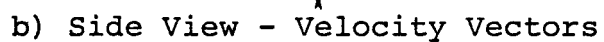
b) Side View - Velocity Vectors

c) Front View - Velocity Vectors

d) Top View - Velocity Vectors

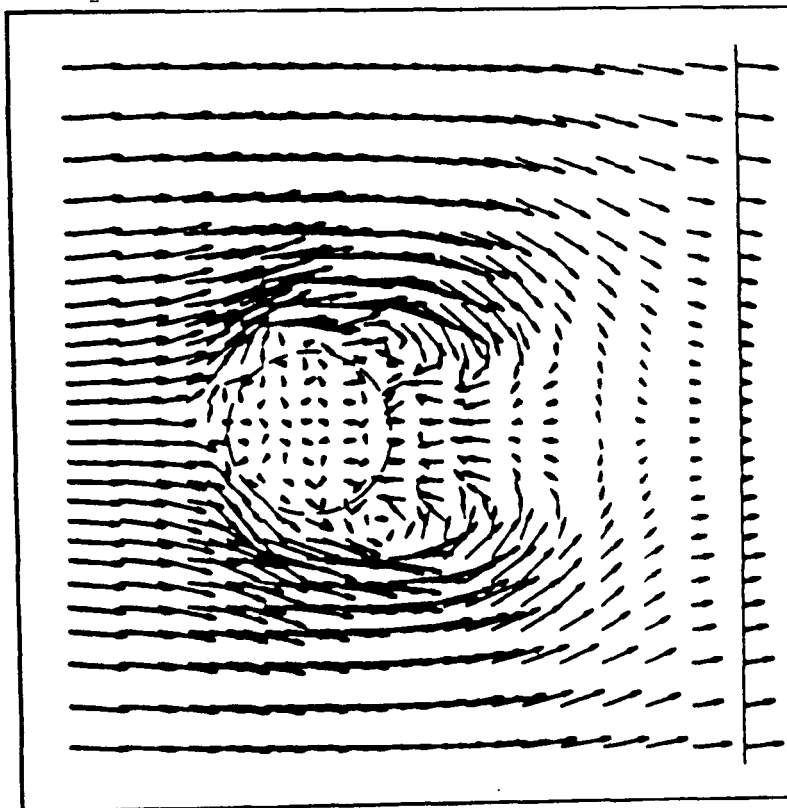Figure 3   Computational Results for the NASA Flat-Plate Plume Model, V_j/V_∞ = 8.0, Specified Upstream Boundary Condition

Computed Results

Experimental Results
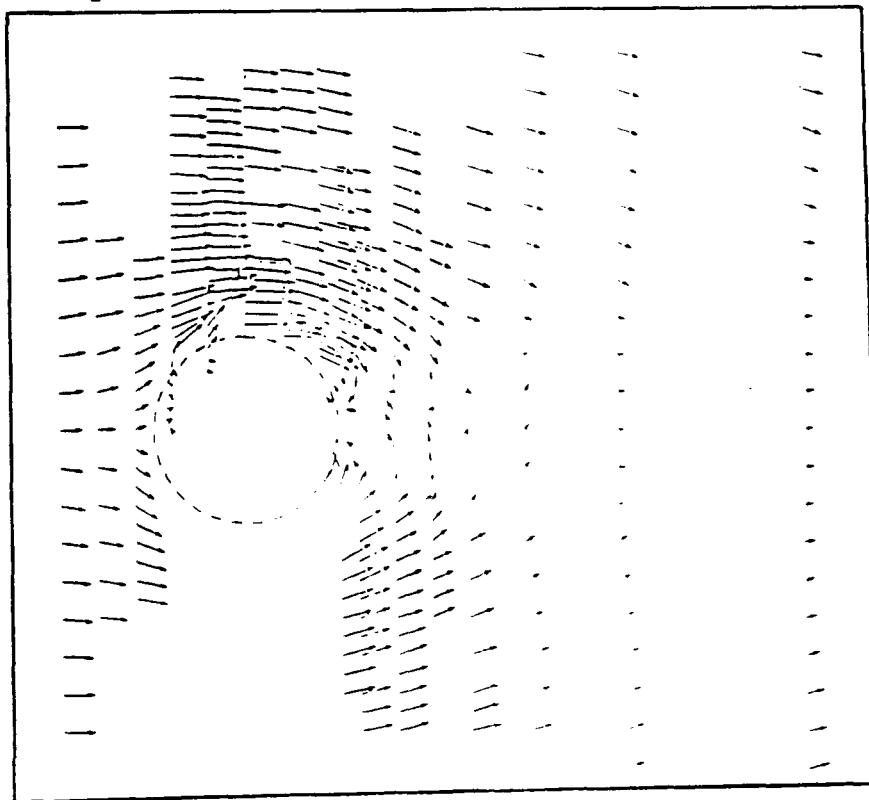
Figure 4   Comparison of Computed and Experimental Results
for the NASA Flat-Plate Plume Model

EXPERIMENTAL TEST CASE

In order to test and validate the computational analysis methodology a suitable test case was necessary. Under contract to General Dynamics, the Israel Institute of Technology (Technion) has tested a series of jet-in-crossflow cases. At a freestream Mach number of 0.3 and a jet Mach number of 1.5, flow field surveys were made at various injection angles. The case with an injection angle of 90° was chosen as the current test case. A schematic of the wind tunnel test set-up is shown in figure 5. Figure 6 shows how data was taken in several planes which are normal to the jet-plume trajectory. Figures 7 and 8 show sample data at a location six jet-exit diameters downstream along the jet-plume centerline.
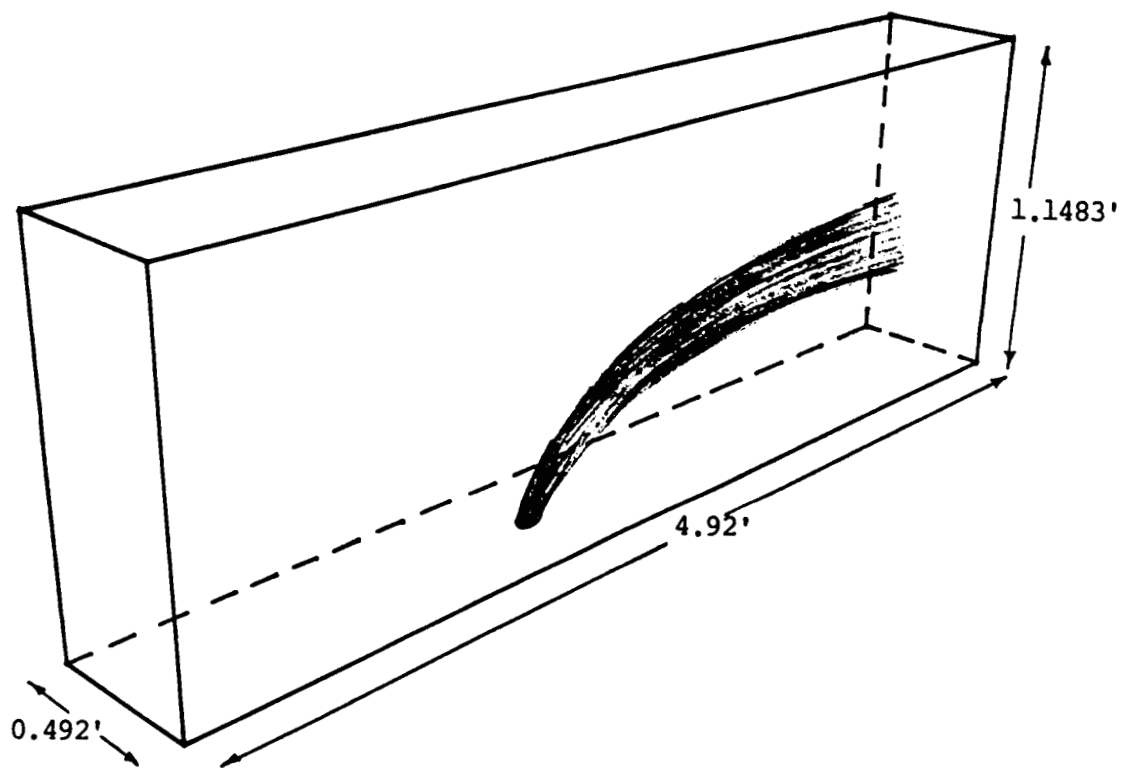
Figure 5 **Schematic of the GD/Technion Jet-in-Crossflow Test Case**

MACH NUMBER

90-DEGREE JET

5 SURVEY PLANES

MACH   0.300
ALPHA   0.00 DEG
GRID   35x17x6

CONTOUR LEVELS
0.30000
0.32500

0.40000
0.45000
0.50000
0.55000
0.60000
0.65000
0.70000

Figure 6   The GD/Technion Jet-in-Crossflow test data was
taken in planes perpendicular to the jet center-
line.

Figure 7   Mach Number Contours for the GD/Technion Jet-in-
Crossflow Wind Tunnel Test at Z/D = 6

COMPUTATIONAL MODEL AND RESULTS

A 70 x 46 x 50 grid (161,000 grid points) was set up for
the General Dynamics/Technion 90 degree jet case. To
alleviate as many boundary condition problems as possible,
the grid included the entire wind tunnel test section. The
grid was basically Cartesian; however, it was clustered
in the vicinity of the jet. Figure 9 shows how the grid was
set-up.

To retain as much generality as possible, a constraint
was placed on the initialization to not assume any type of
jet-plume shape. This was accomplished by initializing the
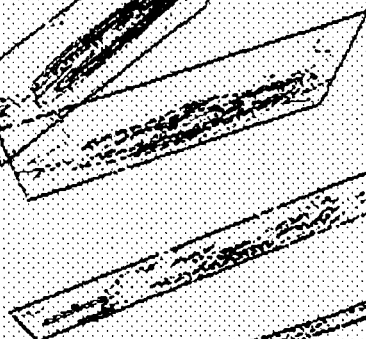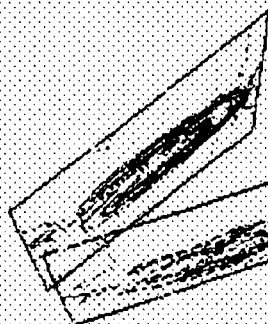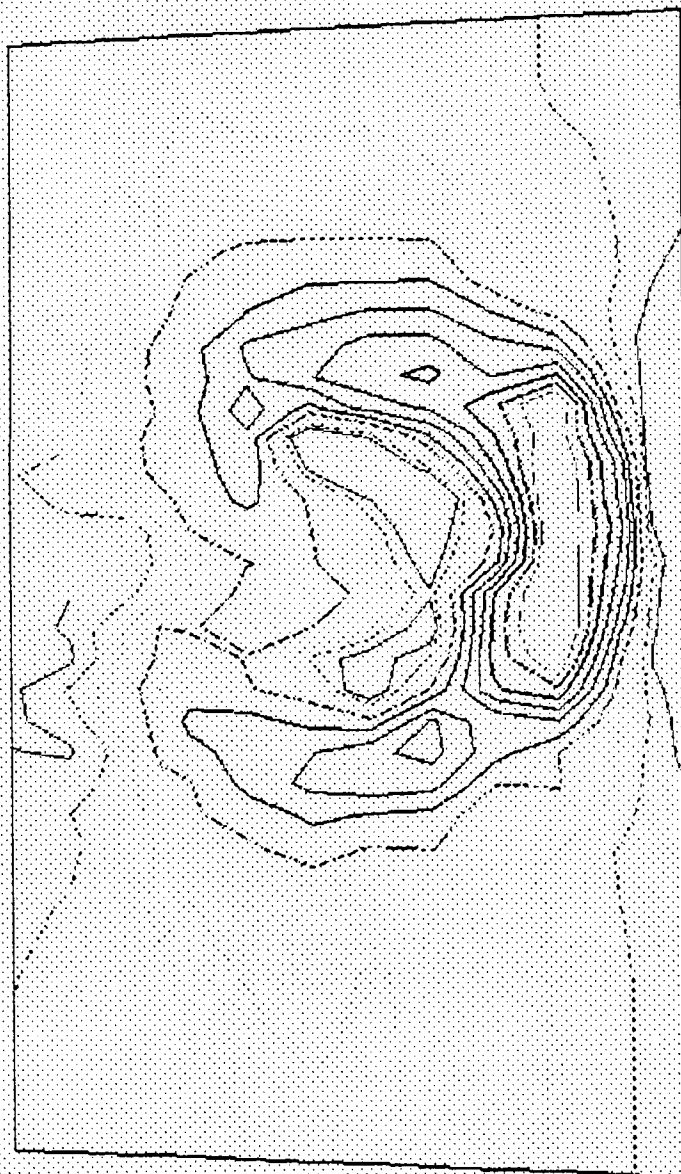jet-plume to be straight and allowing the solution procedure
to turn the jet-plume to its final location.

The freestream and jet flow conditions are given in
Table 1. The inflow boundary conditions were held constant
(with an assumed boundary layer profile) well upstream of the
jet. The outflow boundary conditions were no-gradient in the
flow direction. The lower surface boundary condition was
no-slip and the jet boundary conditions were set at a
constant velocity, density and energy. The side wall
boundary conditions were set to freestream flow and the upper
surface boundary condition was no-gradient in the direction
normal to the boundary. A mixing-length type turbulence

70x16x50   GRID
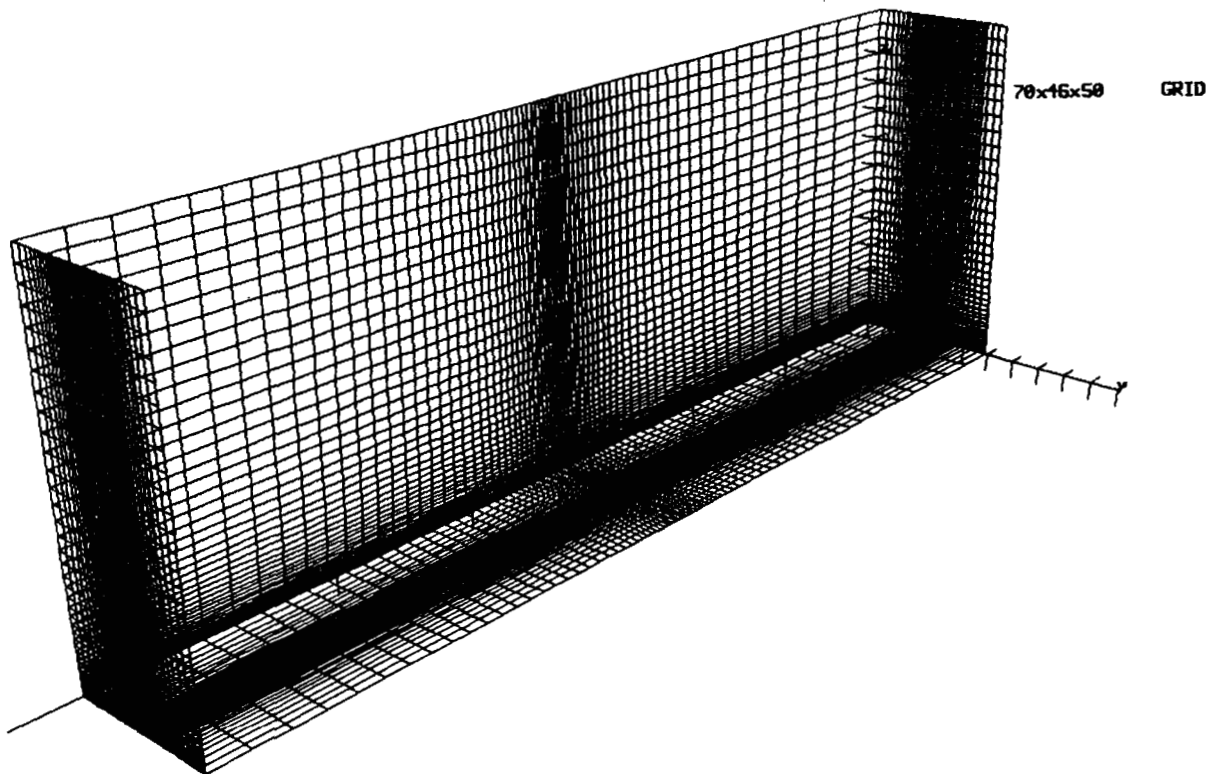
**Figure 9   Computational Grid Set-up for the GD/Technion
Jet-in-Crossflow Test Case**

model was employed where the mixing-length in the region of the jet-plume was assumed to be constant.

TABLE 1 - FLOW CONDITIONS

	Freestream	Jet Exit
Mach Number	0.3	1.5
Static Pressure	2116.2 psf	2116.2 psf
Static Temperature	530.0 deg R	477.2 deg R
Velocity	338.57 fps	1606.2 fps

The solution procedure was executed to ten-thousand iterations on the Numerical Aerodynamic Simulator CRAY-2 super computer. The elapsed solution time was approximately equal to the time required for a particle to travel from one end of the domain to the other at the freestream velocity. Approximately forty-three hours of computer time was used.

Figures 10 through 12 are example results of this analysis. Figure 10 shows velocity vectors in the region of

16

0.300	MACH	
0.00 DEG	ALPHA	
2.07x10**6	Re	
4.46x10**-3	TIME	
70x46x50	GRID	

VELOCITY COLORED BY MACH NUMBER

CONTOUR LEVELS
0.500000
0.600000
0.700000
0.800000
0.900000
1.00000
1.10000
1.20000
1.30000
1.40000
1.50000
1.60000
1.90000
2.00000
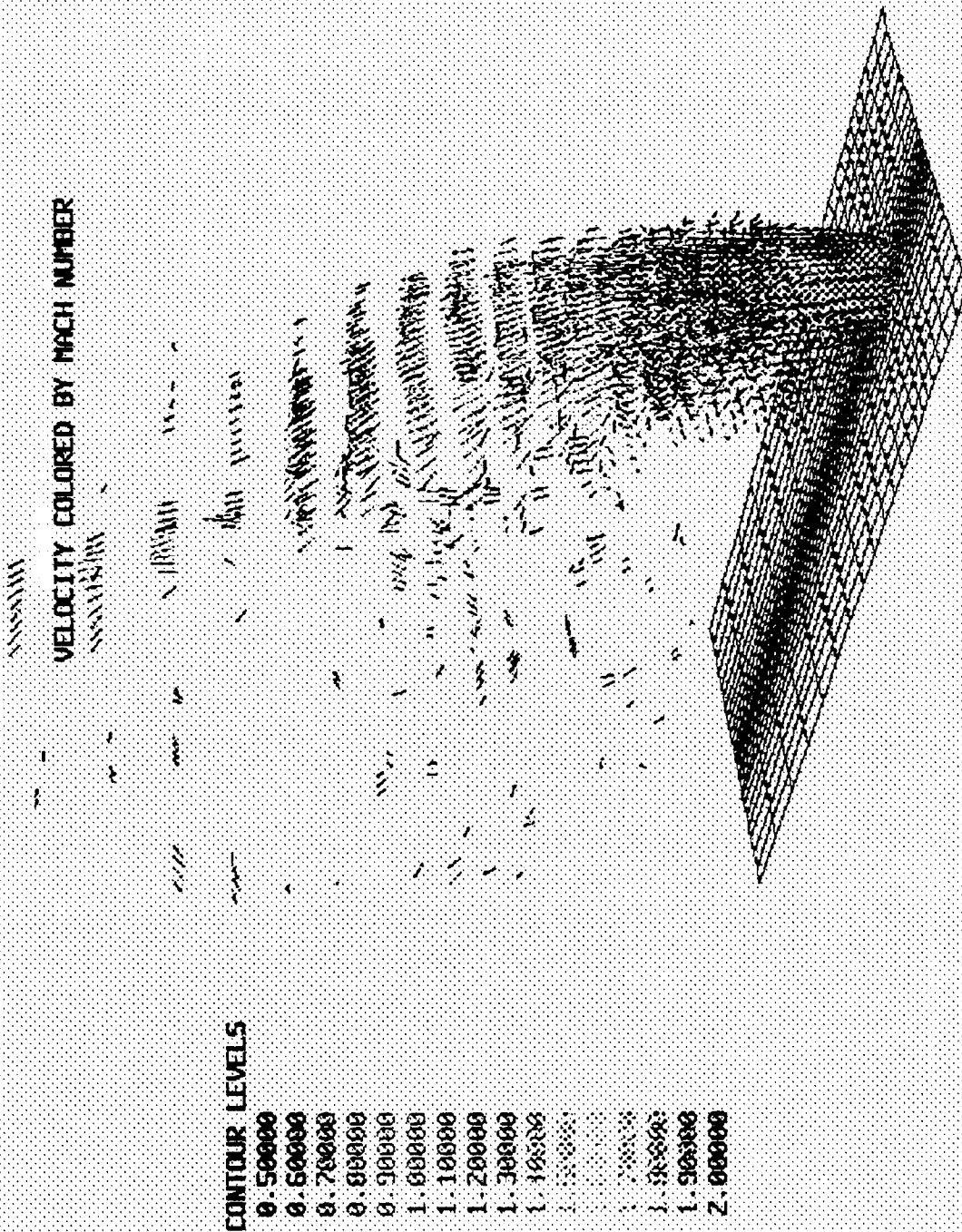
Figure 10   Velocity Vectors Computed for the GD/Technion
Jet-in-Crossflow Test Case

17

Figure 11   Mach Number Contours Computed for the GD/Technion
            Jet-in-Crossflow Test Case
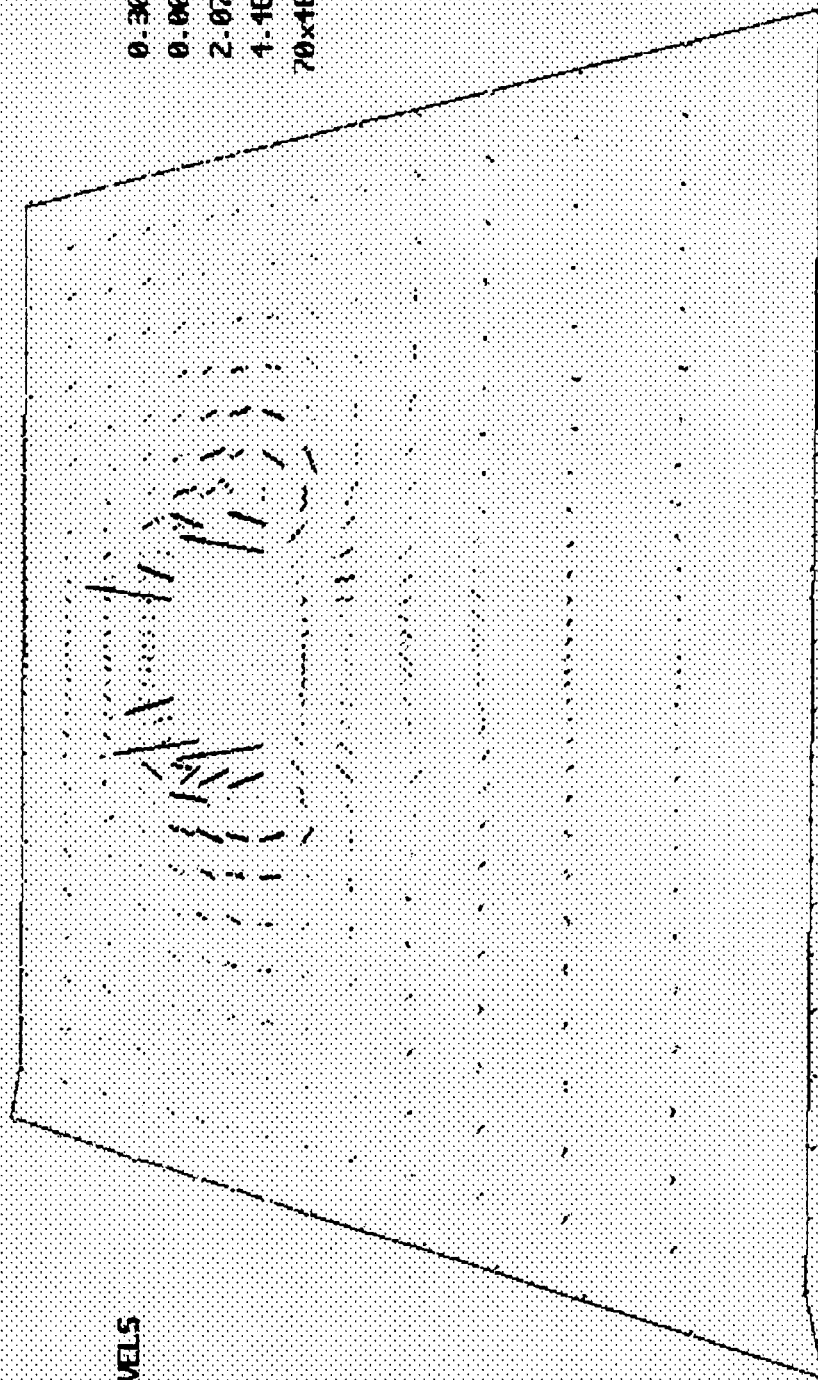
18

VELOCITY COLORED BY MACH NUMBER

MACH   0.300
ALPHA   0.00 DEG
Re   2.07x10**6
TIME   4.16x10**-3
GRID   70x46x50

CONTOUR LEVELS
0.000000
....
0.400000
0.600000
0.800000
1.000000
1.200000

Figure 12   Velocity Vectors Near the Jet Computed for the
             Jet-in-Crossflow Test Case

19

the jet. (Only vectors for points where the Mach number is greater than 0.5 are shown.) Figure 11 shows Mach number contours on a plane cutting through the jet. The jet-plume expansion and mixing are apparent in this figure. The jet initially overexpands and then shocks back down. Figure 12 shows velocity vectors in a plane parallel to the wind tunnel floor. Notice the reversed flow downstream of the jet and the characteristic kidney shape of the jet-plume.

Comparisons with the wind tunnel data were intended but have not been carried out formally because the trajectory of the experimental data was significantly different from the trajectory of the computational data. There are several factors which could be contributors to the discrepancy between the computational and experimental results.

1. The code used in this study was explicit and therefore was very limited in the maximum allowable time-step. For subsonic flows, a good "rule of thumb" for convergence is to allow enough computational time for a particle to traverse the domain at least three times. The current solution has only allowed enough time for one traverse down the domain. Therefore, it is felt that a longer solution time is required.

2. Despite the relatively large number of grid points, grid resolution remains a problem. Since no initial assumptions were made on the plume trajectory, the grid could

not be packed in the region of the jet-plume.

3. Although our ultimate goal would be to not assume anything about the trajectory of the jet-plume, it is becoming apparent that this goal is impractical. Initialization using a good approximate solution could decrease the time to reach a steady result and allow better grid packing in the region of the jet.

4. The turbulent closure of the current case is far from optimum. The mixing-length model used is not appropriate for boundary-layers and lacks the sophistication to adequately simulate the turbulent activity in a jet-in-crossflow. Though it is probably not the dominate problem in the current analysis, turbulent closure should be considered an important part of any jet-in-crossflow analysis. The turbulence level is the major contributor to mixing between the jet and the freestream. This mixing, in turn, plays a major role in determining not only the jet-plume trajectory but the entire flow field definition.

CONCLUSIONS AND RECOMMENDATIONS

The results of this study tend to point out areas of concern rather than provide definitive answers to the numerous problems associated with propulsive-jet flow field analysis. The computational results were not readily comparable to the experimental data because of significant differences between the two data sets. However, much can be learned from this study and applied to future efforts to use the 3-D Navier-Stokes equations to analyze jets-in-crossflow. Based on the results of this study the following recommendations can be made.

1. A time-accurate implicit Navier-Stokes solver should be used instead of an explicit method. This would allow steady solutions to be reached in a reasonable number of time-steps.

2. A grid which adequately resolves the large gradients between the jet and the freestream should be employed. If an approximation to the jet-plume trajectory can be obtained an appropriate grid would be easier to set-up. In addition, a different grid orientation could provide more points in the jet-plume without packing the less interesting outer flow regions. The optimum would be a self adapting grid tied to the flow solver.

3. A lower order approximate solution should be used to initialize the flow field. This would not only provide a more rapid convergence to a steady solution but also provide the basis for grid set-up.

4. Turbulent closure should be accomplished using a higher-order turbulence model. Lakshminarayana (reference 6) suggests using a two-equation model plus an algebraic Reynold's stress model for 3-D flows with curvature rotation and shock waves. Additional investigations need to be carried out in this area.

REFERENCES

1. Adler, D. and Baron, A., "Prediction of a Three-Dimensional Circular Turbulent Jet in Crossflow", <u>AIAA Journal</u>, Vol. 17, No. 2, 1979.

2. Baker, A. J. and Orzechowski, "Prediction of Turbulent Near-Field Evolution of a Jet in Crossflow using a PNS Solver", Report No. NADC-86076-60, Naval Air Development Center, 1986.

3. Shang, J. S., Buning, P. G., Hankey, W. L., and Wirth, M. C., "Performance of a Vectorized Three-Dimensional Navier-Stokes Code on the CRAY-1 Computer", <u>AIAA Journal</u>, Vol. 18, No. 9, 1980.

4. Snyder, P. K., and Orloff, K. L., "3-D LDA Measurements of a Jet in a Crossflow", Preliminary Draft, NASA Ames Research Center, November, 1983.

5. Margason, Richard J., "The Path of a Jet Directed at Large Angles to a Subsonic Freestream," NASA TN-D-4919, NASA Langley Research Center, 1968.

6. Lakshminarayana, B., "Turbulence Modeling for Complex Shear Flows", <u>AIAA Journal</u>, Vol. 24, No. 12, 1986.

1. Report No. NASA CR 177508	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle Propulsive-Jet Flow Field Analysis Using the Three-Dimensional Navier-Stokes Equations		5. Report Date October 1988
		6. Performing Organization Code
7. Author(s) C. L. Reed		8. Performing Organization Report No.
		10. Work Unit No. 505-61-71
9. Performing Organization Name and Address General Dynamics Corporation Fort Worth Division P. O. Box 748 Fort Worth, Texas 76101		11. Contract or Grant No. A30142C (MSB)
		13. Type of Report and Period Covered Contract Report July 1985 – March 1987
12. Sponsoring Agency Name and Address Ames Research Center Moffett Field, CA 94035		14. Sponsoring Agency Code RTOP-505- 61-71

15. Supplementary Notes
Point of contact: Richard Margason, Mail Stop 247-1 Ames Research Center Moffett Field, CA 94035 (415) 694-5033

16. Abstract

    A three-dimensional Navier-Stokes code has been applied to the analysis of flow fields containing propulsive-jets. Specifically, the application was made to a flow field containing a supersonic jet injected at an angle of 90 degree to a subsonic freestream. Although wind tunnel data were available, the computational results were not readily comparable to the experimental data because of significant differences between the two plume trajectories. Reasons for the differences are suggested in the report and include (1) incomplete convergence, (2) inadequate grid resolution in the high gradient regions, and (3) use of a low-order turbulence closure model.

17. Key Words (Suggested by Author(s)) Propulsion, Jet Exhaust, Jet Lift, Navier-Stokes	18. Distribution Statement UNLIMITED Subject category 02 , 64		
19. Security Classif. (of this report) UNCLASSIFIED	20. Security Classif. (of this page) UNCLASSIFIED	21. No. of Pages 25	22. Price* A02