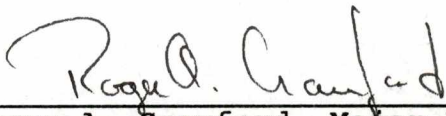


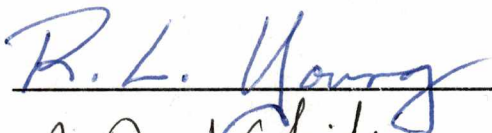
To the Graduate Council:

I am submitting herewith a thesis written by Matthew Brian Prufert entitled "Comparison of Two-Dimensional and Three-Dimensional Computational Fluid Dynamics Solutions for a Vectoring Turbine Engine Exhaust Nozzle." I have examined the final copy of this thesis for form and content and recommend that it be accepted in partial fulfillment of the requirements for the degree of Master of Science, with a major Mechanical Engineering.



Roger A. Crawford, Major Professor

We have read this thesis
and recommend its acceptance:





Accepted for the Council:



Vice Provost
and Dean of The Graduate School

STATEMENT OF PERMISSION TO USE

In presenting this thesis in partial fulfillment of the requirements for a degree at The University of Tennessee, Knoxville, I agree that the Library shall make it available to borrowers under rules of the Library. Brief quotations from this thesis are allowable without special permission, provided that accurate acknowledgment of source is made.

Requests for permission for extended quotation from the reproduction of this thesis in whole or in parts may be granted by the copyright holder.

Signature M. B. Perfect
Date 10 / 31 / 88

Comparison of Two-Dimensional and Three-Dimensional
Computational Fluid Dynamics Solutions for a
Vectoring Turbine Engine Exhaust Nozzle

A Thesis
Presented for the
Master of Science
Degree
The University of Tennessee, Knoxville

Matthew Brian Prufert

December 1988

ACKNOWLEDGEMENTS

The author is very appreciative of the advice and encouragement of Dr. Roger Crawford in completing the Mechanical Engineering program at The University of Tennessee Space Institute (UTSI). Also the efforts of Dr. Ahmad Vakili and Dr. Robert Young are greatly appreciated.

The author extends thanks to the many Sverdrup Technology personnel who helped considerably in their review of the thesis manuscript and technical advice. Special thanks to Mr. F. C. Loper, Dr. G. K. Cooper, Dr. T. W. Swafford, Dr. G. T. Patterson, Dr. C. C. Limbaugh, Mr. J. L. Saunders, Mr. J. R. Parker, Dr. J. C. Adams, and Mr. S. Wehofer.

The calculations provided herein were sponsored by the Arnold Engineering Development Center (AEDC), Air Force Systems Command under a contract with Sverdrup Technology Inc., AEDC Group. The author extends thanks to the United States Air Force and the Arnold Engineering Development Center (AEDC) for the opportunity to attend UTSI. The author is also very appreciative of the permission extended by the managers of Sverdrup Technology, Inc., AEDC Group to attend UTSI.

ABSTRACT

Computational fluid dynamics (CFD) solutions for vectored nozzle flow fields can be used to predict nozzle performance, design ground test facility hardware, and size nozzle instrumentation. The purpose of this research was to compare two-dimensional (2D) and three-dimensional (3D) computational fluid dynamics (CFD) solutions for a two-dimensional vectored exhaust nozzle of rectangular cross section. The motivation for using 2D CFD codes as opposed to 3D CFD codes is that 2D (planar or axisymmetric) CFD simulation results generally have less truncation and round off error, they require less labor and fewer computational resources, they are less subject to interpretive errors; and consequently, results can be provided in a more responsive and timely manner. The parameters that were compared were mass flow, axial gross thrust, and vertical gross thrust. The simulations have been performed at three nozzle pressure ratios, with uniform nozzle inlet boundary conditions. These inlet boundary conditions are based on one-dimensional turbine engine math model predictions. The assumption of uniform nozzle entrance boundary conditions is expected to negate some of the potential for three-dimensional effects;

therefore this investigation represents a "best case" comparison.

The CFD codes (PARC2D and PARC3D) used to model the 2D/CD (two-dimensional convergent-divergent) nozzle flow fields are based on the finite difference form of the Reynolds-averaged Navier-Stokes equations. Usage of the PARC codes is restricted to fluids which have thermally and calorically perfect equations of state and a constant Prandtl number. The codes used to solve these equations incorporated a variant of the implicit approximate factorization algorithm of Beam and Warming. The codes also incorporated a modified Baldwin and Lomax turbulence model.

For the sake of conserving resources, and with the aim of improving simulation responsiveness, ways to obviate a full 3D treatment of 2D/CD nozzle performance predictions were investigated. These methods included:

- 1) Comparisons of 2D and 3D solutions at three nozzle pressure ratios (NPR) for a vectored nozzle configuration of constant area ratio and vector angle.

- 2) An aspect ratio (nozzle width/nozzle height at the nozzle entrance) parametric study to investigate the limits of the more economical 2D CFD performance simulation results.

3) Taking advantage of an assumed plane of symmetry in the flow field in order to reduce the total number of grid points required to resolve the nozzle flow field for 3D simulations.

Predictions for gross thrust, for nozzles with aspect ratios greater than 1, were found to agree within 2 percent for the solutions generated using the PARC2D and the PARC3D codes, indicating the PARC2D code may be useful as an engineering tool for performing engineering tasks associated with vectored nozzles.

Several areas of possible future studies have been identified which will be required in order to determine the capability the PARC codes, in conjunction with flow-field measurements, for the determination of gross thrust within 1 percent of measurement.

TABLE OF CONTENTS

SECTION	PAGE
I. INTRODUCTION.....	1
Background.....	1
Nature of the Flow-Field.....	2
Statement of Problem.....	6
Computational Fluid Dynamics Code.....	7
II. NOZZLE PERFORMANCE CALCULATIONS.....	9
III. COMPARISON TO NOZZLE EXPERIMENTAL DATA.....	14
IV. TWO-DIMENSIONAL SOLUTIONS FOR A VECTORING NOZZLE.....	19
Two-Dimensional Grid.....	20
Grid Packing Considerations.....	24
Boundary Conditions.....	25
Initial Conditions.....	26
Convergence.....	26
Solution For Effective Vector Angle.....	27
V. THREE-DIMENSIONAL SOLUTIONS FOR A VECTORING NOZZLE.....	30
Three-Dimensional Grid.....	31
Grid Packing Considerations.....	32
Boundary Conditions.....	36
Initial Conditions.....	37
Convergence.....	37
VI. 2D AND 3D FLOW-FIELD COMPARISON.....	39
Integrated Values.....	39
Velocity Field and Sonic Line Location Comparison.....	43
VII. NOZZLE ASPECT RATIO PARAMETRIC STUDY.....	51
VIII. SYMMETRY PLANE INVESTIGATION.....	59
IX. CONCLUSIONS.....	63

SECTION	PAGE
X. RECOMMENDED FUTURE EFFORTS.....	65
Flow Field Measurements.....	65
Quantify Secondary Flow Effects.....	66
Transition Duct Modeling.....	66
Leakage and Cooling Flow Modeling.....	67
Nozzle Wall Heat Transfer.....	67
Real Gas Effects.....	68
LIST OF REFERENCES.....	69
APPENDIX.....	73
VITA.....	82

LIST OF FIGURES

FIGURE	PAGE
1. Artist conception of aircraft with a 2D/CD nozzle.....	3
2. 2D/CD nozzles with vectoring and reversing capability.....	4
3. Comparison of predicted discharge coefficients to experimental data.....	16
4. Comparison of predicted sonic line location with experimental data	18
5. 2D grid.....	21
6. Transformation of physical coordinates into computational coordinates.....	22
7. Computational Coordinates.....	23
8. Effective vector angle plotted versus nozzle pressure ratio.....	29
9. Outer surfaces of 3D grid.....	33
10. Three-dimensional grid in computational space.....	34
11. Grid spacing at nozzle exit plane (looking Upstream).....	35
12. Comparison of 2D and 3D memory requirements.....	40
13. Comparison of 2D and 3D solution CPU time requirements.....	41
14. 2D solution (NPR = 3.0) velocity vector plot (top) and isomach plot M=1.0 (bottom).....	44
15. 3D solution (NPR = 3.0) velocity vector plot (top) and isomach plot M=1.0 (bottom).....	45
16. 2D solution (NPR = 6.3) velocity vector plot (top) and isomach plot M=1.0 (bottom).....	46

FIGURE	PAGE
17. 3D solution (NPR = 6.3) velocity vector plot (top) and isomach plot M=1.0 (bottom).....	47
18. 2D solution (NPR = 12.1) velocity vector plot (top) and isomach plot M=1.0 (bottom).....	48
19. 3D solution (NPR = 12.1) velocity vector plot (top) and isomach plot M=1.0 (bottom).....	49
20. Surfaces of grids used in the aspect ratio parametric study.....	52
21. Percent difference curves for mass flow, axial force, vertical force, and effective vector angle.....	55
22. Cross flow plots (aspect ratio = 0.65).....	56
23. Cross flow plots (aspect ratio = 0.32).....	57
24. Cross flow plots (aspect ratio = 0.16).....	58
25. Symmetry plane investigation.....	60

LIST OF TABLES

TABLE	PAGE
1. Axisymmetric nozzle data comparison with PARC code performance predictions for discharge coefficient.....	15
2. Comparison of 2D and 3D solutions for three different nozzle pressure ratios.....	42
3. Aspect ratio parametric study comparison (NPR = 6.3).....	54
4. Symmetry plane investigation (NPR = 6.3).....	61

I. INTRODUCTION

Background

In the past few years significant changes in exhaust nozzle configurations have been developed for use on aircraft which require improved maneuverability. The nozzle configuration changes include two-dimensional, variable geometries with thrust vectoring and reversing capabilities. Two-dimensional nozzles have many claimed advantages for fighter aircraft over conventional axisymmetric nozzles. Two-dimensional nozzles can often be more easily integrated into the airframe to reduce drag, Ref. [1]. In addition, two-dimensional nozzles can be adapted to thrust vectoring and/or thrust reversing with less weight penalty than proposed axisymmetric vectoring/reversing nozzles Ref. [2]. An advantage of thrust vectoring and thrust reversing capabilities is reduced takeoff and landing rolls. This short takeoff and landing capability is important for operation from icy or bomb damaged runways Ref. [3]. Thrust vectoring and reversing may also be used during flight to increase aircraft maneuverability and low speed flight capabilities Ref. [4]. Thrust reversing can modulate the thrust with no spool-up delays, and thrust vectoring on aircraft with aft mounted nozzles is an effective

alternative pitch control. Because aerodynamic control surfaces are usually sized for low speed operation (low dynamic pressure), they usually provide more control power than is required at higher speeds. Thrust vectoring generates control moments that are uncoupled from the airframe aerodynamics and thus provides the aircraft designer with the opportunity to eliminate or reduce the size of conventional control surfaces Ref. [5]. An artists conception of an aircraft incorporating a two-dimensional vectoring and reversing nozzle is shown in Figure 1.

Nature of The Flow-Field

There are many types of two-dimensional nozzles under investigation. One of the most common of these are two-dimensional convergent divergent (2D/CD) nozzles. Typical 2D/CD nozzles are depicted in Figure 2. When operating in the supersonic mode the turbine engine exhaust gas enters the nozzle subsonically and accelerates to transonic speeds as it passes through the nozzle throat. The gas then leaves the nozzle as a supersonic jet which interacts with the ambient flow. Within 2D/CD nozzles the flow is confined by the nozzle sidewalls. There are three-dimensional flow field effects due in part to the

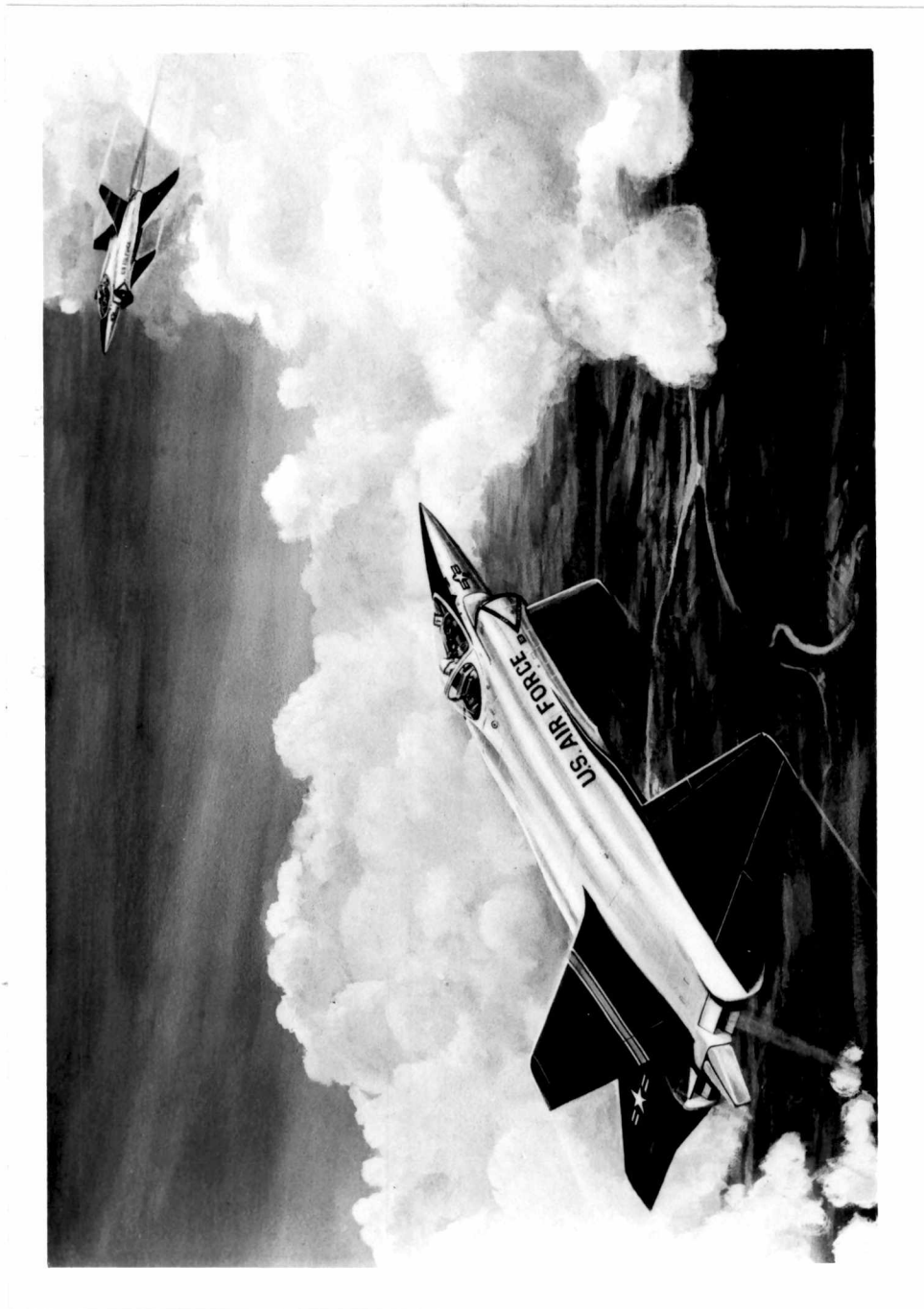


Figure 1. Artist conception of aircraft with a 2D/CD nozzle.

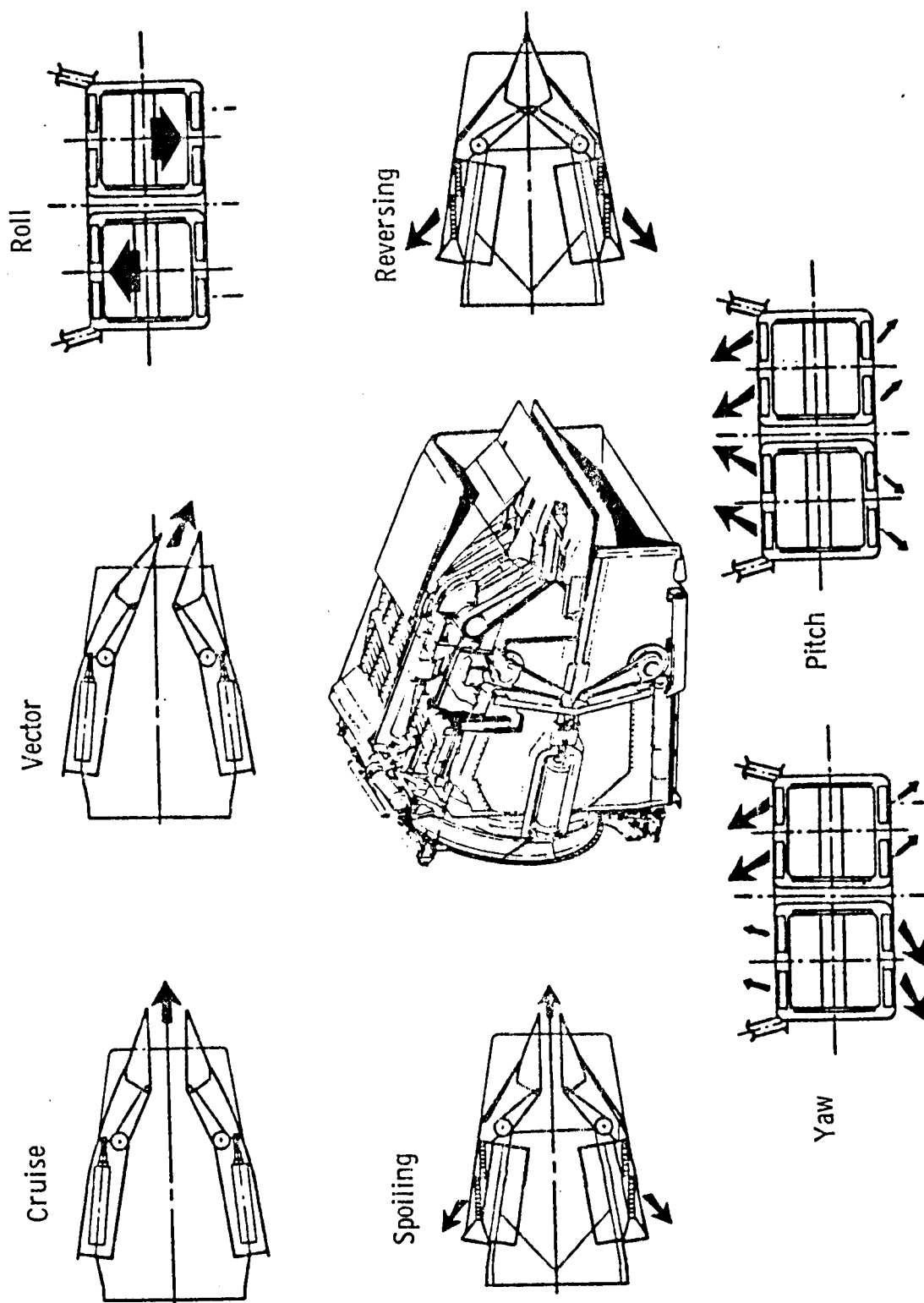


Figure 2. 2D/CD nozzles with vectoring and reversing capability.

viscous boundary layers on the nozzle walls. The influence of the boundary-layer is a function of the wetted surface area, thus the nozzle aspect ratio. Once the flow exits the nozzle, however, it is no longer confined and the resulting expansion can affect the flow within the nozzle. These three-dimensional effects are small near the nozzle exit but become more significant as the distance from the exit increases. Nozzle performance predictions are based on flow within the nozzle. For subsonic nozzle flow fields the nozzle internal flow is highly dependent on the external flow. For supersonic nozzle flow fields the flow within the nozzle is less dependent upon the external flow, however information is still transferred back up into the nozzle through the subsonic portion of the boundary layer.

Flow field vectoring by 2D/CD nozzles can be readily accomplished by differential positioning of the divergent flaps. Utilizing this method can result in a wide range of vectored performance characteristics and losses. As the nozzle is vectored, pressure gradients are introduced into the flow field to accommodate the change in geometry. The severity of the pressure gradients introduced into the flow is dependent on the divergent flap angles, nozzle pressure ratio (NPR), area ratio, and vector angle. The presence of pressure gradients within the flow field presents the potential to induce cross-flows Ref. [2].

Statement of Problem

The analysis of two-dimensional thrust reversing and thrust vectoring nozzles requires considerable understanding of the nozzle internal flow fields. Although internal performance prediction techniques for axisymmetric nozzles are well-established, this is not true of advanced two-dimensional nozzles. Prior applications of CFD for the prediction of axisymmetric nozzle performance have capitalized on an assumed axis of symmetry in order to reduce the complexity of both the measurement systems and computational algorithms. In contrast to conventional axisymmetric turbine engine exhaust nozzles, two-dimensional nozzles produce inherently three-dimensional exhaust gas flow fields. The purpose of this research was to compare 2D and 3D CFD solutions for a vectored 2D/CD turbine engine exhaust nozzle. The motivation for using 2D codes as opposed to 3D codes is that 2D (planar or axisymmetric) CFD simulation results generally have less truncation and round off error, they require less labor and fewer computational resources, they are less subject to interpretive errors; and consequently, results can be provided in a more responsive and timely manner. For the sake of conserving resources, and with the aim of improving simulation responsiveness, ways to

obviate a full 3D treatment of two-dimensional nozzle performance predictions were investigated. These methods included: 1) comparisons of 2D and 3D solutions at three different nozzle pressure ratios; 2) an aspect ratio parametric study to investigate the limits of the more economical 2D CFD performance simulation results; 3) taking advantage of an assumed plane of symmetry in the flow field in order to reduce the total number of grid points required to resolve the nozzle flow field for 3D simulations.

The results of these investigations are presented in sections IV through VIII.

Computational Fluid Dynamics Code

The CFD code chosen for this study is a version of the Pulliam-Steger code (The NASA Ames Research Center "ARC" code) which has been extensively modified to produce an applications-oriented general purpose Navier-Stokes code (The PARC code) with emphasis on propulsion testing. The code contains a variant of the implicit approximate factorization algorithm of Beam and Warming applied to the compressible Navier-Stokes equations expressed in curvilinear coordinates. Details on the mathematics of the PARC code are presented in appendix A.

Both two-dimensional and three-dimensional versions of the PARC code (PARC2D and PARC3D) are currently being used to solve many nozzle flow field problems. Reddy and Harloff Ref. [6] have reported validation of the PARC3D code for a variety of complex and high speed flow configurations, including a high area ratio hypersonic nozzle of rectangular cross section. Cooper and Phares Ref. [7] have applied the PARC2D code to a trumpet and a conical nozzle with a throat Reynolds number of 12,000. The ground tested nozzles had expansion ratios of 362 and 400 and exhausted into low pressures of $3.8\text{E-}07$ to $6.2\text{E-}03$ psia. They concluded that PARC2D code provided reasonable flow fields for the cases studied. The PARC2D code was also used to model a flow field representative of a Boeing convergent nozzle for which extensive experimental data exist Ref. [8]. Agreement between the PARC code predictions and the Boeing nozzle test data for discharge coefficient was found to be within one percent. The comparison of computed results with the Boeing data will be presented in section III.

II. NOZZLE PERFORMANCE CALCULATIONS

The exhaust nozzle is a very important component of the overall engine propulsion system. The sensitivity of the net thrust delivered by the engine to the nozzle performance is higher than any other engine component, Ref. [9]. For this reason, it is extremely important to obtain the highest possible nozzle performance with due consideration for nozzle cost, weight, complexity, and reliability.

A nozzle is a device which converts enthalpy into kinetic energy by accelerating the working fluid, Ref. [10]. The thrust coefficient is defined as the ratio of the actual nozzle gross thrust to ideal available gross thrust and thus represents a measure of nozzle efficiency. The nozzle thrust coefficient includes losses due to the following:

- (1) Friction (momentum loss due to wall friction in the nozzle)
- (2) Swirl (momentum loss due to non-thrust producing velocity components at the nozzle exit plane)
- (3) Expansion (loss due to mismatch of the nozzle exit pressure with ambient)
- (4) Leakage
- (5) Cooling air throttling and mixing loss

The axial gross force, F_x was calculated by integrating the exit axial momentum flux obtained from the PARC code solutions, plus the "pressure-area" term which results from integration of the difference between the computed nozzle exit and ambient pressures acting over the projected nozzle exit area.

$$F_x = \int_0^{A_{ex}} \frac{(mv_x)_e}{A_{ex}(g_c)} dA_{ex} + \int_0^{A_{ex}} (P_e - P_{amb}) dA_{ex} \quad (1)$$

Where

F_x = Axial gross force.

$(mv_x)_e$ = Axial momentum flux at the nozzle exit.

P_e = Static pressure at the nozzle exit.

P_{amb} = Ambient pressure at the nozzle exit.

A_{ex} = Projected exit area normal to the axial velocity component.

Similarly, the vertical gross force F_y and the side gross force (force acting normal to the x-y plane) F_z were obtained by integrating the above equation for the vertical and side force terms acting over their respective areas.

For the 2D CFD simulations the resultant gross thrust F_R was calculated using the integrated axial and vertical gross force for the 2D solutions as shown below:

$$F_R = (F_x^2 + F_y^2)^{1/2} \quad (2)$$

For the 3D CFD simulations the resultant gross thrust was calculated using all three force components.

$$F_R = (F_x^2 + F_y^2 + F_z^2)^{1/2} \quad (3)$$

The resultant thrust coefficient (CFGR) is defined as the ratio of the resultant thrust to ideal gross thrust (FGI):

$$CFGR = F_R / FGI \quad (4)$$

The ideal thrust was calculated as:

$$FGI = W_j V_i \quad (5)$$

Where

W_j = nozzle gas flow rate calculated using the PARC code.

V_i = ideal velocity.

For the convergent-divergent nozzle simulations the ideal velocity, V_i , was based on isentropic expansion to ambient pressure using given total conditions. For the convergent nozzle simulations the ideal velocity was limited to sonic velocity.

$$V_i = M_e (\gamma R T_{se} g_c)^{1/2} \quad (6)$$

Where

M_e = Mach number at nozzle exit.

γ = Exhaust gas specific heat ratio based on
representative exhaust gas temperature and
fuel air ratio.

R = Specific gas constant.

T_{se} = Static temperature at the nozzle exit.

The discharge coefficient was also calculated in order to make comparisons between PARC code predictions and published axisymmetric convergent nozzle discharge coefficients. The published discharge coefficients presented are the ratio of the measured nozzle mass flow to ideal nozzle flow. The predicted discharge coefficients based on the PARC code solutions are the ratio of the calculated nozzle mass flow to the ideal nozzle flow.

$$\text{Discharge Coefficient} = W/W_i \quad (7)$$

Where

W = Nozzle gas flow rate calculated using
the PARC code.

W_i = Isentropic gas flow rate.

The isentropic gas flow rate was calculated from the
conditions at the nozzle throat as follows:

$$W_i = \sqrt{g_c \gamma / R} M_{th} \sqrt{(1 + ((\gamma - 1) / 2) M_{th}^2)} A_{th} P_{th} / \sqrt{T_{tth}} \quad (8)$$

Where

M_{th} = Mach number at the nozzle throat.

A_{th} = Throat area of nozzle.

P_{th} = Static pressure at the nozzle throat.

T_{tth} = Total temperature at the nozzle throat.

M_{th} , P_{th} and T_{tth} were determined from the isentropic
flow relations, known pressure ratio, area ratio, and
specific heat ratio.

III. COMPARISON TO NOZZLE EXPERIMENTAL DATA

Prior to the application of the PARC code to 2D/CD nozzle flow fields, an investigation was performed in order to evaluate the PARC code's ability to simulate axisymmetric nozzle flow fields. This could be accomplished because extensive experimental data were available for the comparisons. In this section, the effect of viscosity on nozzle discharge coefficients, thrust coefficients, and sonic line location was determined using the PARC code. The effect of viscosity was studied by comparison of flow field solutions generated with the PARC2D code, excluding the viscous flux terms (inviscid solution), and including the viscous flux terms (viscous solution). The grids used in this investigation were developed in a grid refinement study in which the nodal density was increased until the viscous solutions stopped changing with respect to nodal density. This methodology was used for a Boeing 40 degree convergent nozzle and a 25 degree convergent nozzle. The calculated discharge coefficients were compared with the experimental values published by Boeing and are shown in Table 1 and Figure 3. Discharge coefficients obtained from the viscous solutions were found to agree with the Boeing data within 0.6 percent, whereas the inviscid solutions produced coefficients

Table 1. Axisymmetric nozzle data comparison with PARC code performance predictions for discharge coefficient. Ref. [8].

	Measured	Inviscid	Viscous
<u>Boeing 40 degree axisymmetric convergent nozzle NPR=2</u>			
Discharge coefficient	0.867	0.880	0.873
Thrust coefficient*	--	0.884	0.874
<u>Boeing 40 degree axisymmetric convergent nozzle NPR=4</u>			
Discharge coefficient	0.915	0.924	0.917
Thrust coefficient	--	0.936	0.928
<u>Boeing 25 degree axisymmetric convergent nozzle NPR=2</u>			
Discharge coefficient	0.920	0.926	0.915
Thrust coefficient	--	0.930	0.918
<u>Boeing 25 degree axisymmetric convergent nozzle NPR=4</u>			
Discharge coefficient	0.949	0.959	0.946
Thrust coefficient	--	0.971	0.957

*Thrust measurements for the Boeing nozzles were not published.

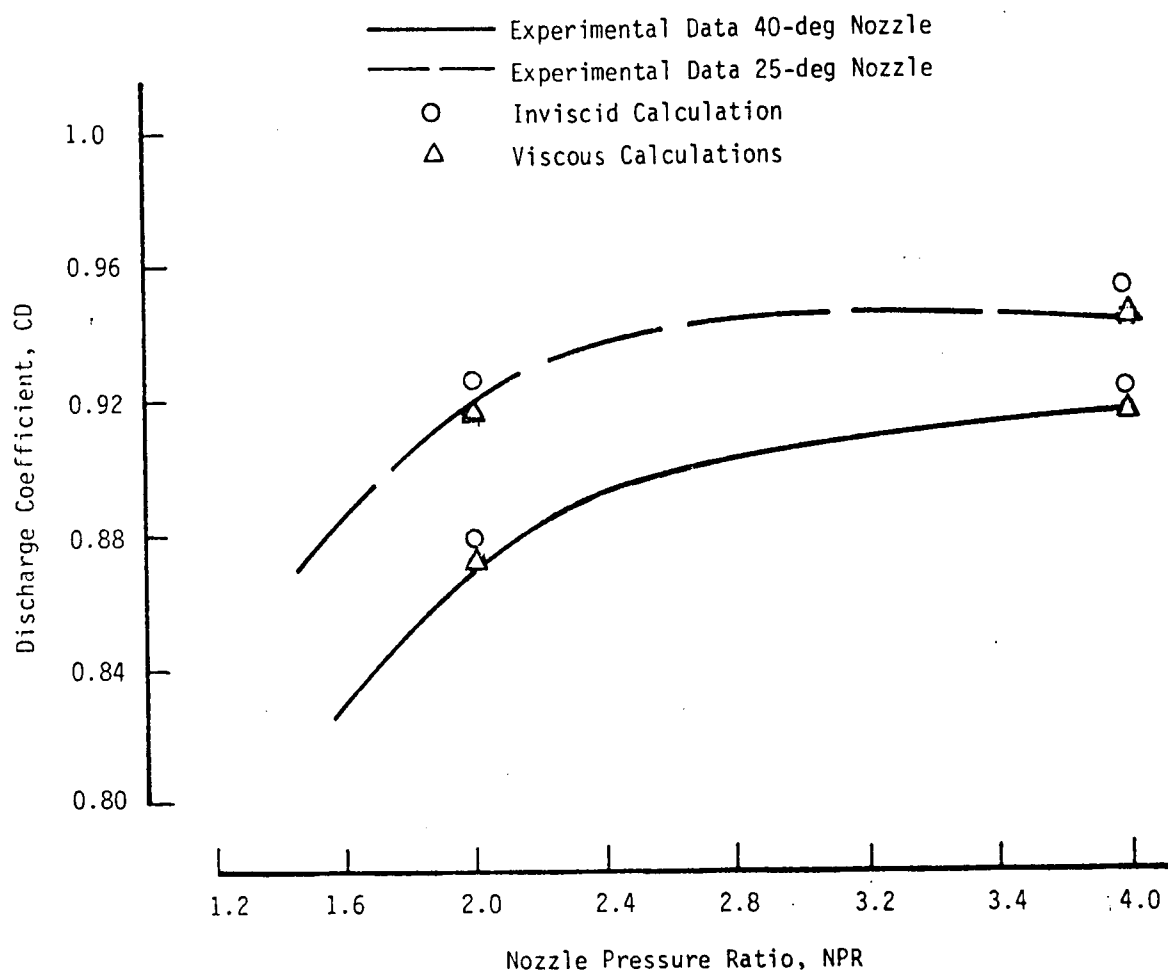


Figure 3. Comparison of predicted discharge coefficients to experimental data.

which were within 1.3 percent of those measured. A comparison between measured and calculated sonic line locations for both a viscous and inviscid solution is shown in Figure 4. The results indicate that the viscous solution more accurately models the flow field; therefore viscous solutions were utilized in subsequent 2D/CD nozzle analysis.

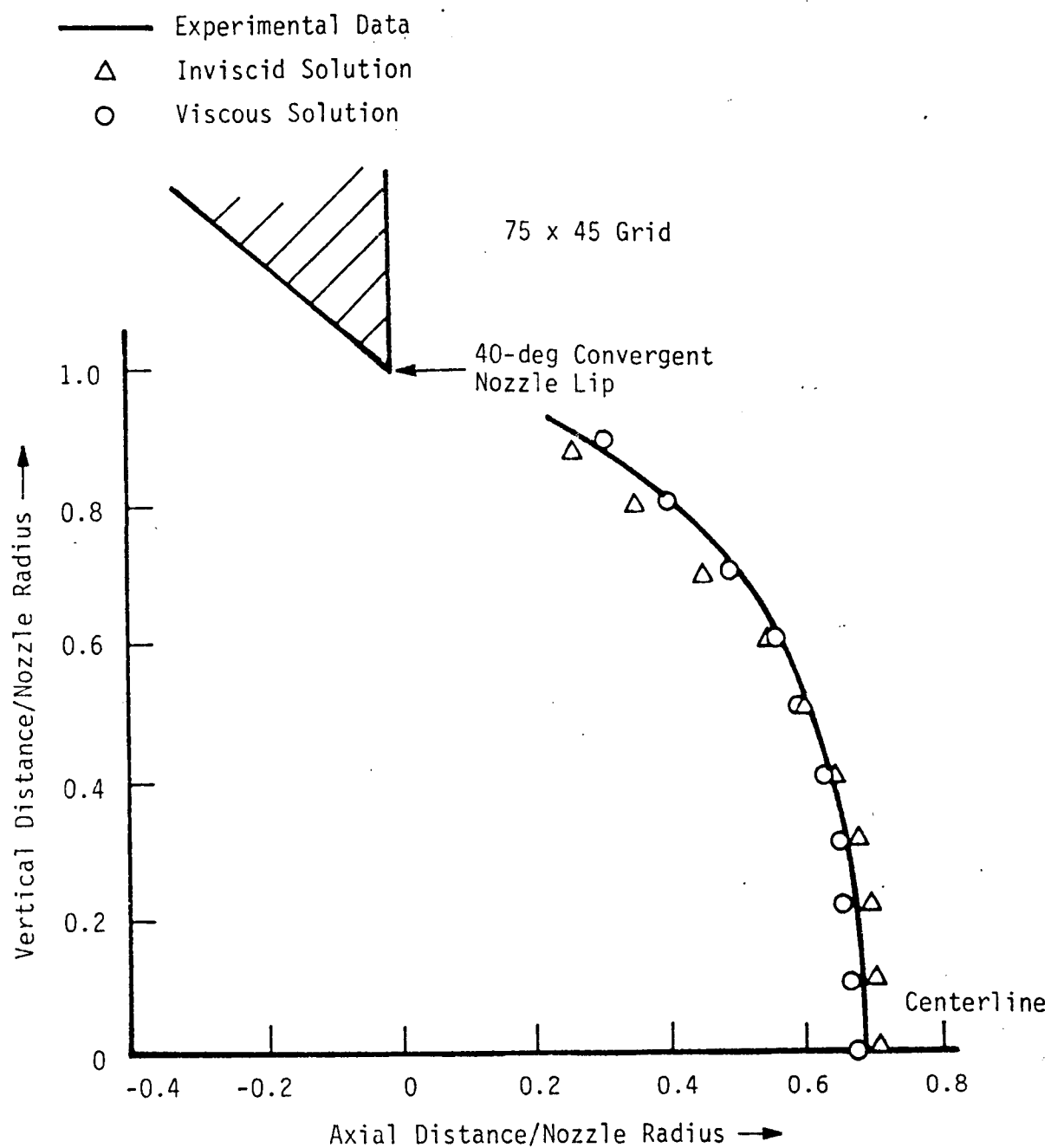


Figure 4. Comparison of predicted sonic line location with experimental data.

IV. TWO-DIMENSIONAL SOLUTIONS FOR A VECTORING NOZZLE

Three different 2D/CD nozzle solutions corresponding to three different nozzle pressure ratios were modeled using the PARC2D code. A premise of the 2D solutions is that the flow field is planar and that there are no velocity components in the direction normal to the plane. These solutions are analogous to those which would exist if the 2D/CD nozzle were of infinite depth in the direction normal to the 2D plane. For this investigation the following assumptions were made in order to simplify the analysis:

1. The flow was steady and could be approximated as a calorically and thermally perfect gas with a constant specific heat ratio (γ).
2. The turbulence could be modeled algebraically using a modified Baldwin-Lomax model.
3. Nozzle inlet plane total pressure and total temperature profiles were uniform for the nozzle entrance boundary condition.
4. No heat transfer occurred along the nozzle walls (adiabatic).
5. The nozzle leakage, and injected nozzle cooling airflow were not modeled.
6. The nozzle sidewalls were not modeled.

Two-Dimensional Grid

Before a numerical solution of the Navier-Stokes equations can be performed using the PARC2D code, a suitable distribution of grid points for the computational domain must be obtained. The 2D grid that was used in this study is shown in Figure 5. The grid was built by subdividing the nozzle geometry and the surrounding ambient region into four individual patches, where one of the patches was internal to the nozzle and the remaining three were external to the nozzle. The indices of the grid are most easily shown by considering the grid in computational space. Figure 6 shows the physical coordinates of the nozzle transformed into computational coordinates. Internal to the nozzle, 80 grid lines were used in the horizontal direction (referred to as the J indices) and 60 were used in the vertical direction (referred to as the K indices). The external regions above and below the nozzle were generated using 80 grid lines in the J index and 20 in the K index (referred to as ambient regions 1 and 2 in Figure 7). Early in this investigation it was decided not to perform calculations in the ambient regions 1 and 2 because the modeling of external flow around the nozzle was not within the scope of this research.

The portion of the grid for the ambient region downstream of the nozzle exit (referred to as ambient region

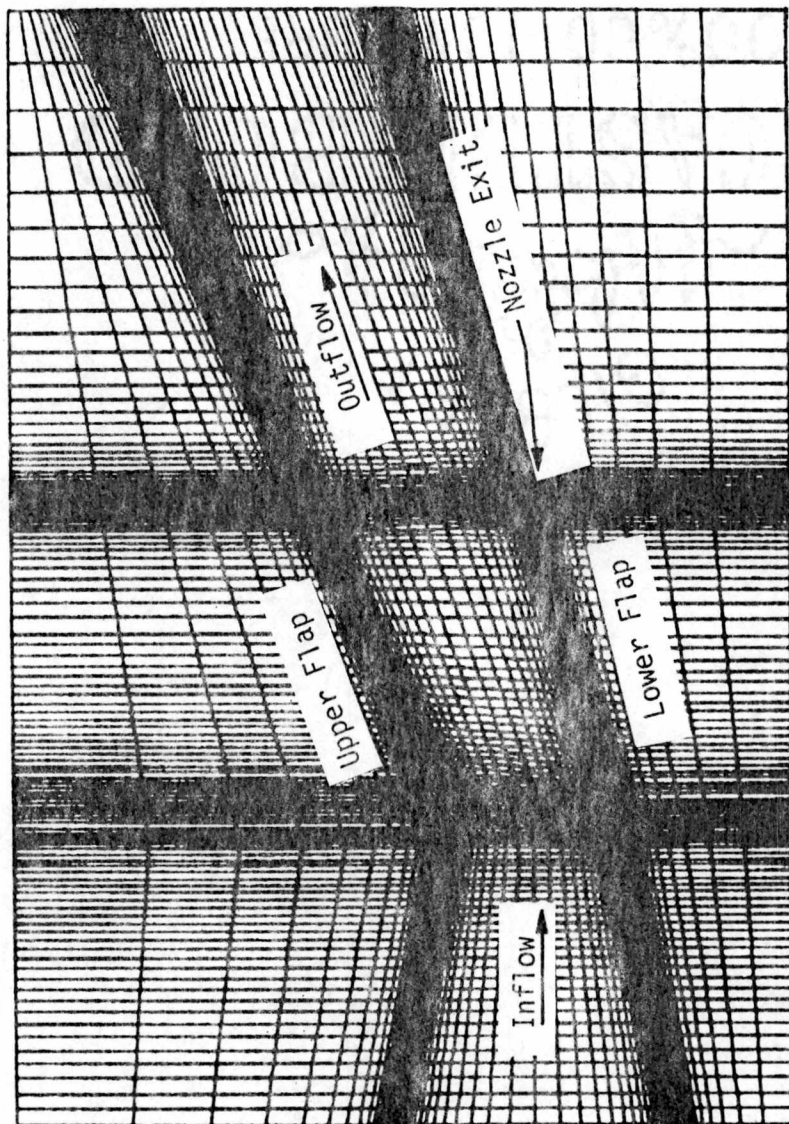


Figure 5. 2D grid.

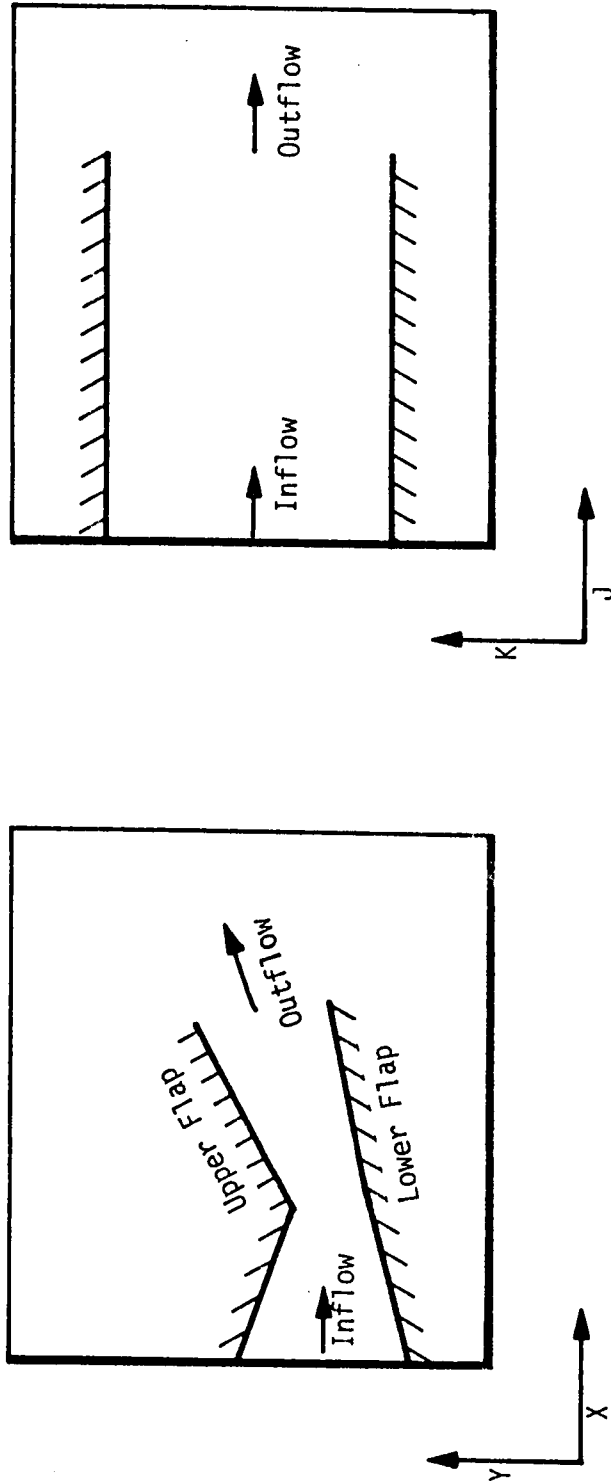


Figure 6. Transformation of physical coordinates into computational coordinates.

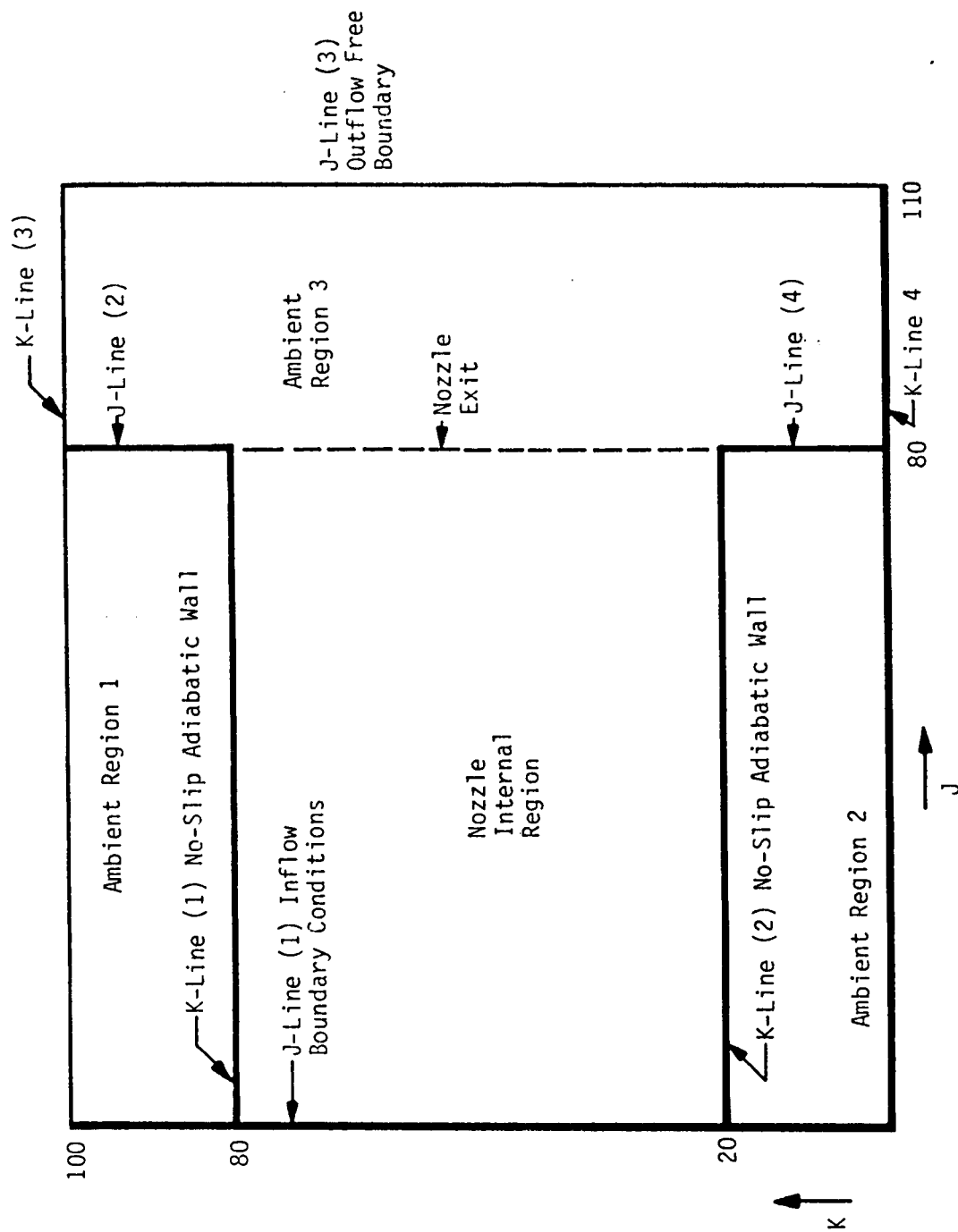


Figure 7. Computational coordinates.

3 in Figure 7) was generated using 30 grid lines in the J index and 100 grid lines in the K index. The total number of grid points used for the 2D solutions was the product of the 80 J index grid lines used inside the nozzle times 60 K index grid lines used inside the nozzle plus 30 J index grid lines times 100 K index grid lines used in ambient region 3 which equals a total of 7,800 grid points used in the 2D simulations.

Grid Packing Considerations

Internal to the nozzle, grids were packed exponentially at the top and bottom no-slip walls of the nozzle with the first grid point 0.05 inches away from the no-slip wall. This dense packing in the region close to the wall was chosen in order to resolve the boundary layer on the upper and lower surfaces of the nozzle (this distance was based on previous grid refinement studies performed with the PARC code and was found to resolve the boundary layer when the solutions were analyzed). Grids were also densely packed in the nozzle throat region and the nozzle exit plane region in order to provide high resolution of these regions with high flow gradients. In general, better computational results, (in terms of accuracy and convergence rates) can be obtained if the grid is constructed such that flow gradients can be

resolved, and the grid varies smoothly with minimal skewness Ref. [12]. External to the nozzle, grids were packed densely in the free stream region where the jet plume meets the stagnant ambient air.

Boundary Conditions

The PARC codes are designed so that specification of boundaries and boundary conditions is done entirely through inputs to the program. The boundary conditions selected in order to solve this particular problem consisted of six free boundaries and two adiabatic no-slip walls. A free boundary is defined as any boundary segment through which the fluid flow can freely pass and is not a known supersonic inflow boundary or a constant mass flux boundary. The boundary shown in Figure 7 as J-Line(1) was the inflow free boundary condition for the nozzle problem. For this inflow free boundary condition the total pressure and total temperature were specified. The boundary conditions represented as J-line(2), J-line(3), J-Line(4), K-Line(3), and K-Line(4) were considered outflow free boundaries.

A no-slip adiabatic wall boundary condition was chosen to represent the upper and lower nozzle walls shown as K-Line(1) and K-Line(2) in Figure 7. This boundary condition

stipulates that all velocity components and the normal gradients of pressure and temperature are set to zero.'

Initial Conditions

The PARC codes are designed so that every application of the code is treated as if a partially converged flow field was available for input; that is, execution of these codes always requires a restart file. The initial conditions that were used to start the flow simulations consisted of setting the initial gas dynamic properties of all the nodes within the nozzle to that of the nozzle entrance boundary conditions for the first iteration. Similarly, for the region outside the nozzle, all of the gas properties were set equal to those of the outflow boundary conditions. These initial conditions provided an adequate starting point for the simulations.

Convergence

For the three different 2D solutions performed, the flow field within the nozzle appeared to converge within 1000 iterations of the PARC code; however, the flow field outside the nozzle (which contains the jet plume and the free stream mixing region) required approximately 3000

additional iterations to achieve the same level of convergence as that of the internal nozzle flow field. The region where the plume meets the free stream ambient flow was expected to be the most difficult portion of the flow-field to resolve because the boundary conditions are not as well specified as for the nozzle internal flow.

Two criteria were used to determine if convergence could be assumed: 1) Plots of isomachs (constant Mach number contours) were compared after every 1000 iterations to qualitatively discern if the flow field had stopped changing, and 2) The sum of the root mean square of the individual residuals of mass flux, momentum fluxes, and energy flux, was required to be less than 0.00001. All three 2D solutions converged within 4000 iterations and each required slightly less than 900 seconds CRAY-XMP CPU time. Detailed results of the three 2D solutions are presented in section VI.

Solution For Effective Vector Angle

The two-dimensional CFD calculations were performed with the PARC code in order to explore the possibility of producing effective vector angles that deviate from the geometric vector angle. The geometric vector angle is defined as the angle between the engine axial center-line

and the average divergent flap angle. The effective vector angle is defined from the following equation.

$$\text{effective vector angle} = \arctan \left[\frac{\text{vertical force}}{\text{axial force}} \right] \quad (9)$$

The results of this CFD study show that the effective vector angle is larger than the geometric vector angle at the low nozzle pressure ratio case simulated. Effective vector angles larger than the geometric vector angles have been previously reported for a 2D/CD nozzle tested at low nozzle pressure ratios Ref. [3]. The results also show a trend where the effective vector angle becomes increasingly smaller than the geometric vector angle with increasing nozzle pressure ratio, as shown in Figure 8. Analysis of these results indicate that the flow-field profiles at the nozzle exit are dependent on the nozzle exit pressure ratio. At higher nozzle pressure ratios the exhaust gas has a higher associated velocity; thus, more momentum than for the lower nozzle pressure ratio cases, therefore increasing the nozzle pressure ratio makes the flow more difficult to turn.

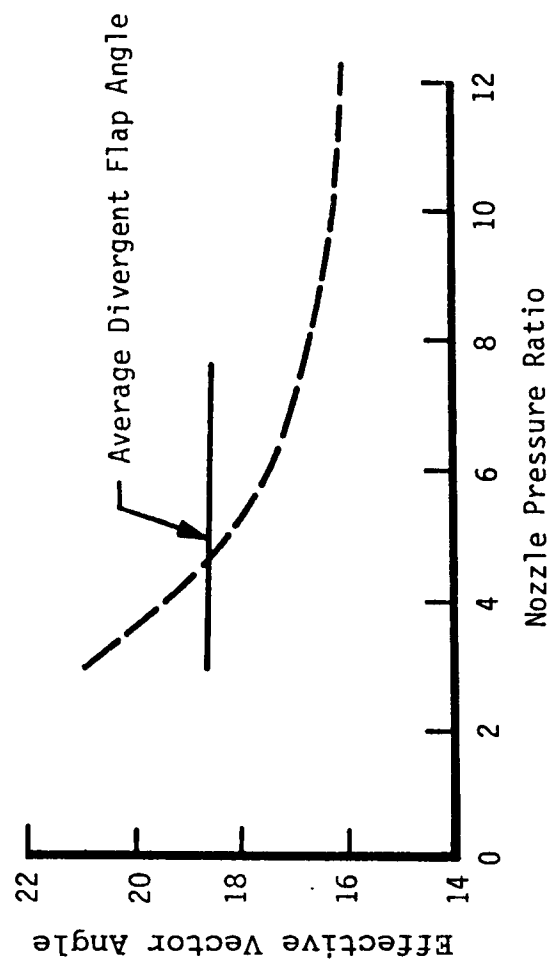


Figure 8. Effective vector angle plotted versus nozzle pressure ratio.

V. THREE-DIMENSIONAL SOLUTIONS FOR A VECTORING NOZZLE

The same three nozzle pressure ratios that were modeled using the PARC2D code were modeled using the PARC3D code. Some of the same assumptions and simplifications that were used for the 2D simulations were made for the 3D simulations. Specifically for the 3D simulations the following assumptions were made:

1. The flow was steady and could be approximated as a thermally and calorically perfect gas with a constant specific heat ratio (γ).

2. The turbulence could be modeled algebraically using a modified Baldwin-Lomax model.

3. Nozzle inlet plane total pressure and total temperature profiles were uniform for the nozzle entrance boundary condition.

4. No heat transfer occurred along the nozzle walls (adiabatic).

5. The nozzle leakage, and injected nozzle cooling airflow were not modeled.

In contrast to the 2D simulations the nozzle sidewalls were modeled for the 3D solutions using no-slip adiabatic walls.

Three-Dimensional Grid

In the 2D grid a generous portion of the "ambient" field was modeled to complete the flow domain. It was determined from the 2D simulations that a high degree of flow-resolution beyond the nozzle exit did not significantly influence computed performance parameters because the flow was predominantly supersonic (based on the 2D simulations 97 percent of the area in the nozzle exit plane contained supersonic flow). At the nozzle exit region the subsonic boundary layer was expected to be thin resulting in a restricted region within which information could pass upstream. It was also determined from the 2D simulations that the regions outside the nozzle where the jet plume meets the stagnant ambient air required over twice as many iterations to converge as the flow inside the nozzle required to converge. These facts were capitalized upon in the generation of the 3D grid. For the 3D problem the ambient region was treated by extending "inviscid walls" downstream of the nozzle exit plane. The reason for doing this was to reduce the number of grid points required for the computation such that the calculations could be performed within the limits of the CRAY-XMP main memory size of 2 million words (MWDS). The largest 3D grid that

the PARC3D code can execute within the CRAY-XMP main memory is one that contains less than approximately 64,000 nodes. The grid that was used in this study used 35 grid lines in the J index, 40 grid lines in the K index and 40 grid lines in the L index resulting in a total of 56,000 nodes as opposed to the 7,800 nodes used in the 2D simulations. The outer surfaces of the 3D grid is shown in Figure 9. A representation of the grid after transformation into computational space is shown in Figure 10.

Grid Packing Considerations

Internal to the nozzle, grid points were packed exponentially at the top and bottom no-slip nozzle walls, (as shown in Figure 11), with the first grid point 0.05 inches away from the no-slip wall. Likewise across the nozzle, grids were packed exponentially at the left and right nozzle sidewalls with the first grid point 0.05 inches away from the no-slip wall. This dense packing in the region close to the wall was chosen in order to resolve the boundary layers on the nozzle walls. Grids were also densely packed in the nozzle throat region and the nozzle exit region in order to resolve these important domains. It

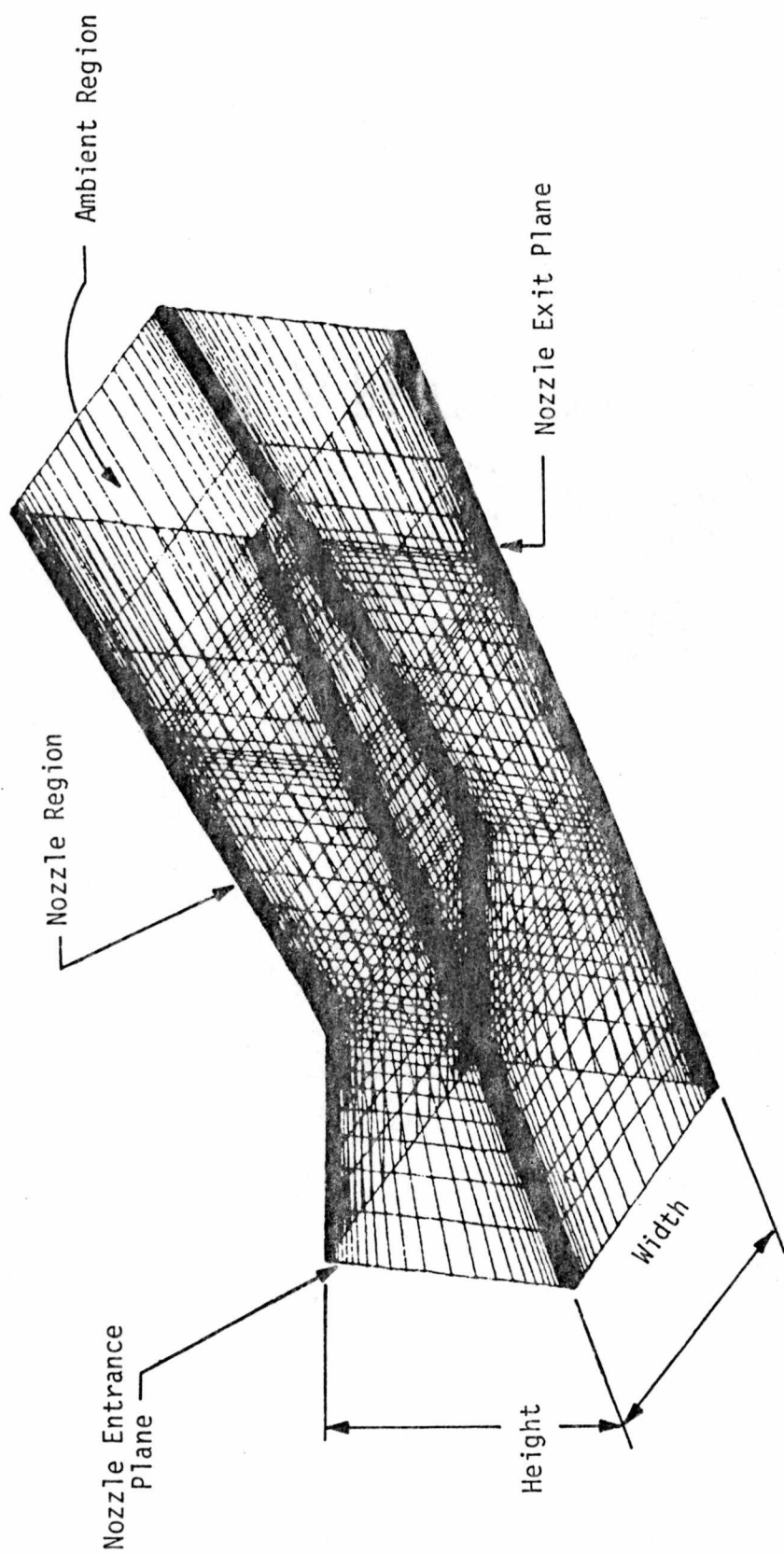


Figure 9. Outer surfaces of 3D grid.

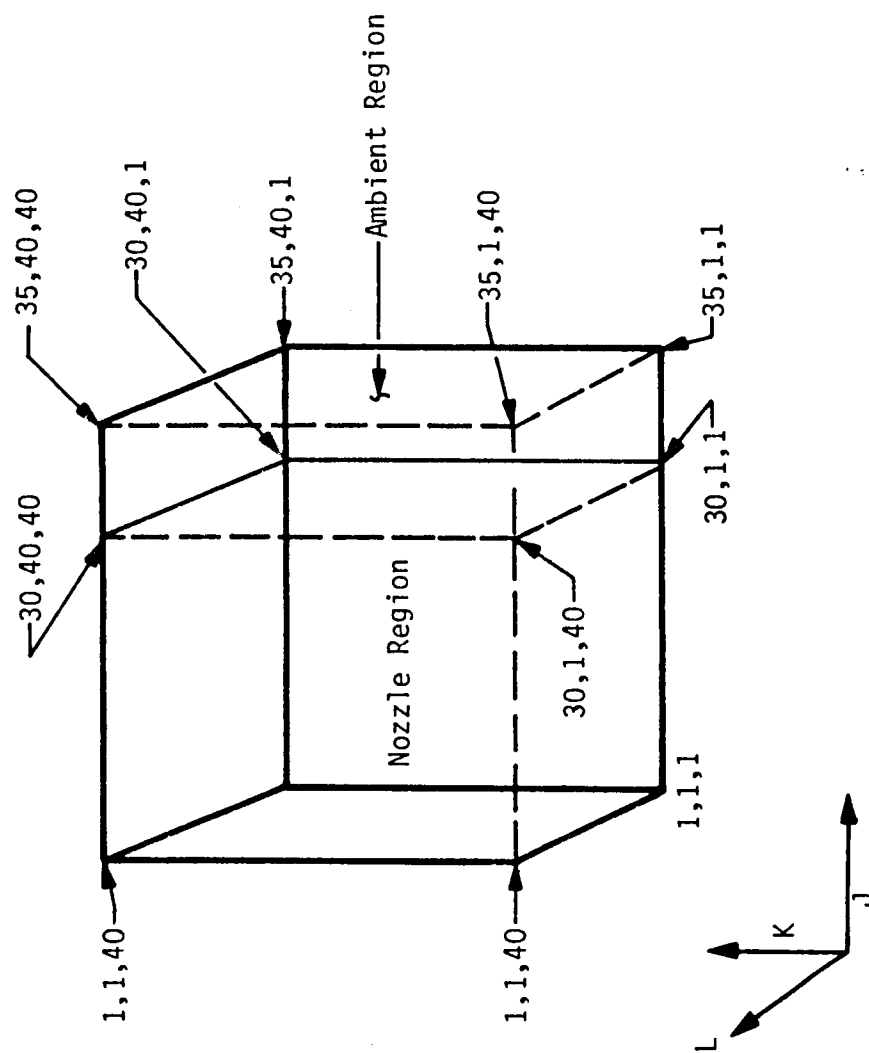


Figure 10. Three-dimensional grid in computational space.

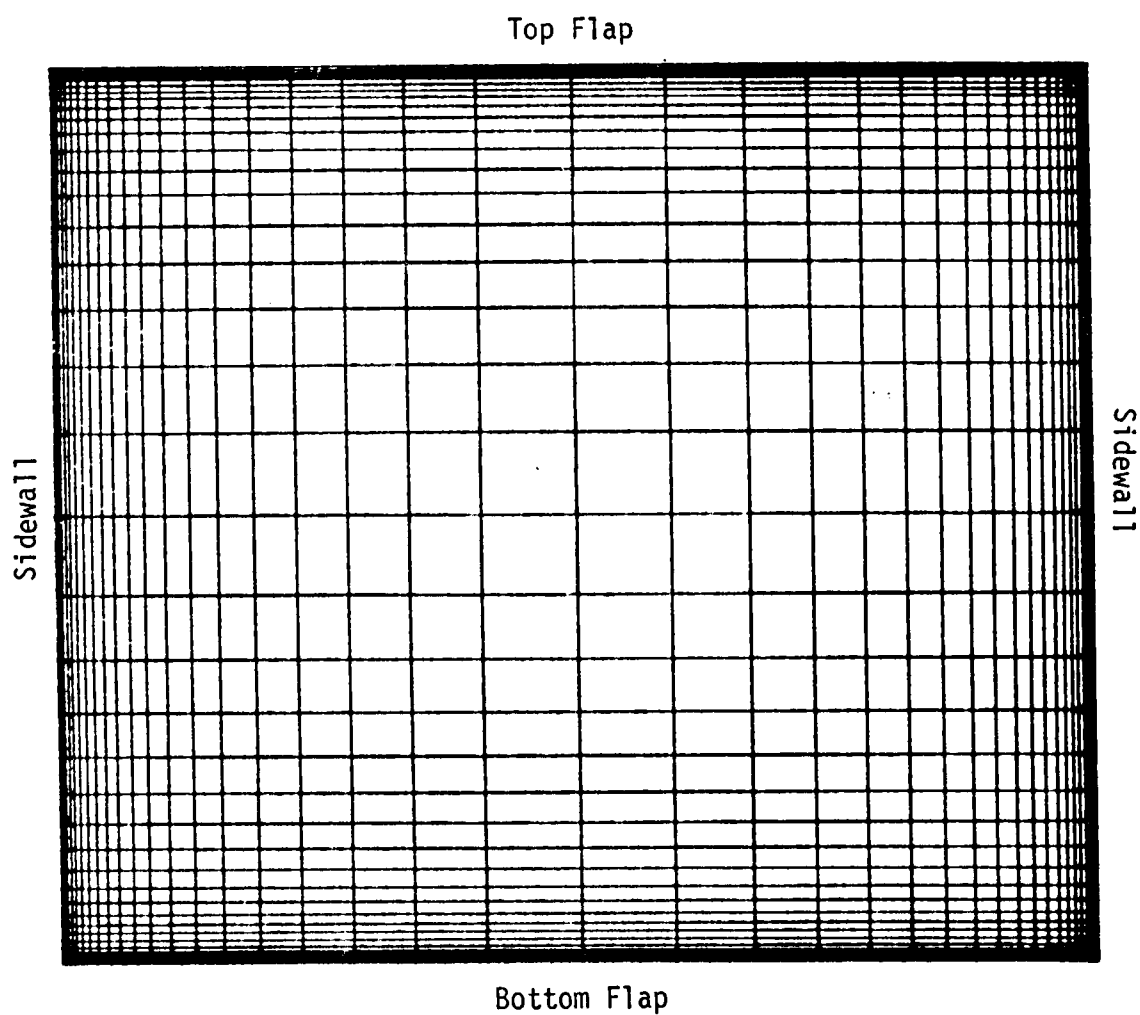


Figure 11. Grid spacing at nozzle exit plane (looking upstream).

should be noted that the 2D solutions permitted a higher degree of resolution (4,800 grid points) inside the nozzle than an analogous 2D plane extracted from the 3D grid (containing 1,200 grid points).

Boundary Conditions

The boundary conditions used were similar to those used in the 2D solutions with the exception that very little of the ambient region outside the nozzle was modeled, and slip walls were incorporated in order to reduce the total number of grid points used. The boundary conditions consisted of four no-slip adiabatic walls, four slip walls, one inflow free boundary and one outflow free boundary. For 3D solutions these boundaries are represented as planes in the computational space. At the inflow boundary the total pressure and temperature are held constant. On the no-slip adiabatic walls the velocity components and the normal gradients of pressure and temperature are set to zero. For the slip walls the density, pressure, and velocity magnitude at the slip wall boundary are determined by extrapolation from points adjacent to the boundary, while the velocity direction is determined from the wall tangency condition. At the outflow free-boundary the flow was supersonic. For this boundary condition the static pressure is prescribed,

density, and velocity were extrapolated from the upstream conditions.

Initial Conditions

The initial conditions that were used to initiate the 3D simulations consisted of setting the initial gas dynamic properties at all the nodes within the nozzle to those of the nozzle entrance boundary conditions for the first iteration. Similarly, for the region outside the nozzle, all gas properties were set equal to those of the outflow boundary conditions. These initial conditions provided an adequate starting point for the simulations.

Convergence

Although the ambient region was simplified for the 3D simulations, these calculations still required considerably more time to reach convergence than the 2D simulations. All three nozzle pressure ratios simulated converged within 4,000 iterations requiring approximately 13,500 CRAY-XMP CPU seconds. For the nozzle parametric study, the number of iterations required for the 3D simulations was cut in half (approximately 2000 iterations per solution). This was

accomplished by using a converged solution from the initial 3D simulations as the initial condition.

VI. 2D AND 3D FLOW-FIELD COMPARISON

A comparison was made of the 2D and 3D simulated performance results for the three nozzle pressure ratios simulated. Figures 12 and 13 show the required computer resources that were necessary to generate the converged 2D and 3D solutions. The memory requirements for the 3D simulations presented were found to be over 3 times as high as those required for the 2D simulations presented. The amount of CPU time required for each of the 3D simulations presented was found to be over 15 times as high as the amount of CPU time required for the analogous 2D simulations.

Integrated Values

The integrated values of mass flow, axial force, vertical force and effective vector angle for both the 2D and 3D simulations are shown in Table 2. The 2D solutions shown did not model the nozzle sidewalls as opposed to the 3D solutions where the nozzle sidewalls were modeled by using no-slip wall boundary conditions. For the case with $NPR=3.0$, the agreement between the 2D and 3D solutions for resolved force and effective vector-angle was within 1.43 percent. For the cases with $NPR=6.3$ and 12.1 the agreement for

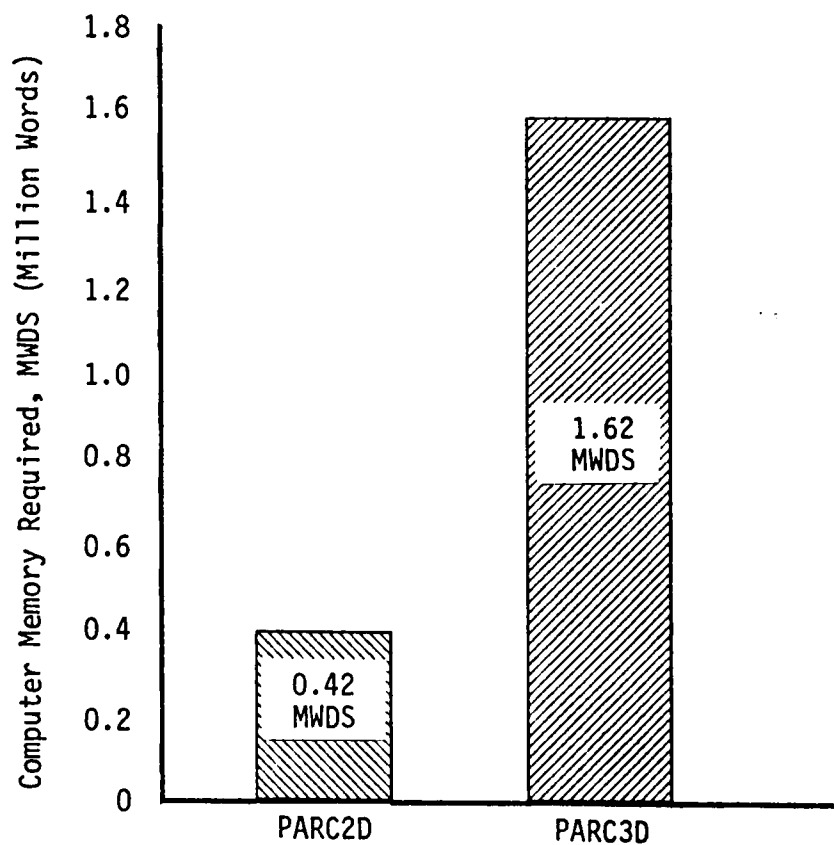


Figure 12. Comparison of 2D and 3D memory requirements.

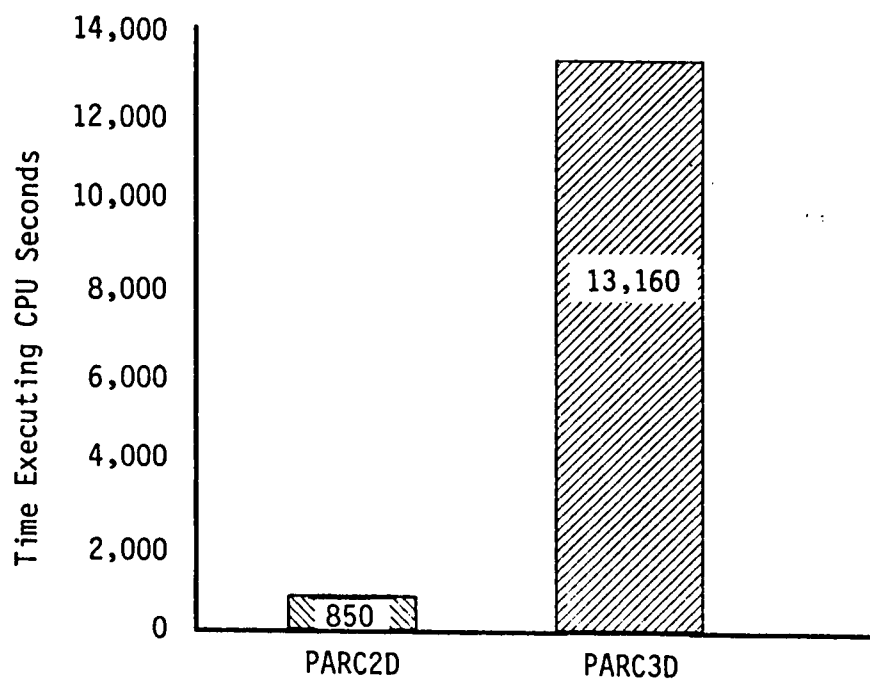


Figure 13. Comparison of 2D and 3D solution CPU time requirements.

Table 2. Comparison of 2D and 3D solutions for three different nozzle pressure ratios.

	2D 110x100	3D 35x40x40	$\left \frac{2D-3D}{2D} \right * 100$
<u>Case 1</u>			
<u>NPR=3.0, P7=20.95 psia, T7=3512 deg.R, Pamb=7.0 psia, Tamb=523 deg.R</u>			
Mass flow, lbm/s	118.0	117.5	0.42
Axial force, lbf	11601.0	11440.0	1.39
Vertical force, lbf	4552.0	4476.0	1.67
Side force, lbf	--	-1.0	--
Effective vector	21.4	21.4	0.0
Resolved force, lbf	12462	12285	1.43
CFGR	0.975	0.961	1.43
<u>Case 2</u>			
<u>NPR=6.3, P7=21.3 psia, T7=3519 deg.R, Pamb=3.37 psia, Tamb=523 deg.R</u>			
Mass flow, lbm/s	117.6	118.1	-0.43
Axial force, lbf	14852.0	14748.0	0.7
Vertical force, lbf	4704.0	4720.0	-0.34
Side force, lbf	--	-0.2	--
Effective vector	17.6	17.7	-0.57
Resolved force, lbf	15579	15485	0.61
CFGR	0.975	0.969	0.61
<u>Case 3</u>			
<u>NPR=12.1, P7=20.86 psia, T7=3451 deg.R, Pamb=1.73 psia, Tamb=537 deg.R</u>			
Mass flow, lbm/s	116.2	116.5	-0.26
Axial force, lbf	15947.0	15858.0	0.56
Vertical force, lbf	4599.0	4594.0	0.11
Side force, lbf	--	-1.0	--
Effective vector	16.1	16.2	-0.62
Resolved force, lbf	16597	16510	0.52
CFGR	0.948	0.943	0.52

NPR= Total pressure at nozzle entrance/ambient pressure.

P7- Pressure at nozzle entrance., T7- Temperature at nozzle entrance.

Pamb- Ambient pressure., Tamb- Ambient temperature.

resolved force and effective vector angle was within 0.7 percent. It should be noted that for the NPR=3.0 case, the nozzle was overexpanded; whereas for the cases where NPR=6.3 and 12.1, the nozzle was underexpanded. It is possible that the three dimensional flow effects are more significant for the overexpanded configuration (NPR=3.0 case), thus contributing to the larger difference between the 2D and 3D CFD simulations.

Velocity Field and Sonic Line Location Comparison

Sonic line plots and velocity field plots are presented in Figures 14 through 19. These plots qualitatively illustrate differences between the 2D and 3D CFD simulations. The flow field plots for the 3D CFD simulations correspond to the mid-z plane half way into the nozzle flow field. It should be noted that the 2D problems (Figures 14, 16 and 18) are not scaled the same as their 3D counterparts (Figures 15, 17, and 19). The 2D results, with a higher resolution grid than the 3D counterpart (especially in the ambient region) show some interesting qualitative properties not apparent in the 3D solutions. An over-expanded nozzle flow field with oblique shocks originating off of the nozzle's upper and lower divergent flaps is shown in Figure 14. A slightly under-expanded nozzle flow field (i.e. static

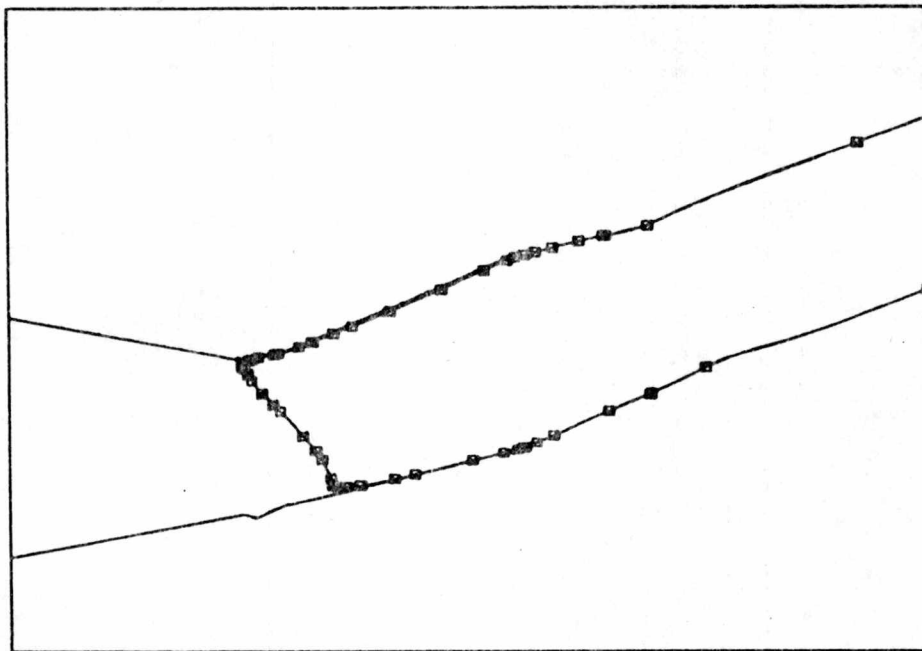
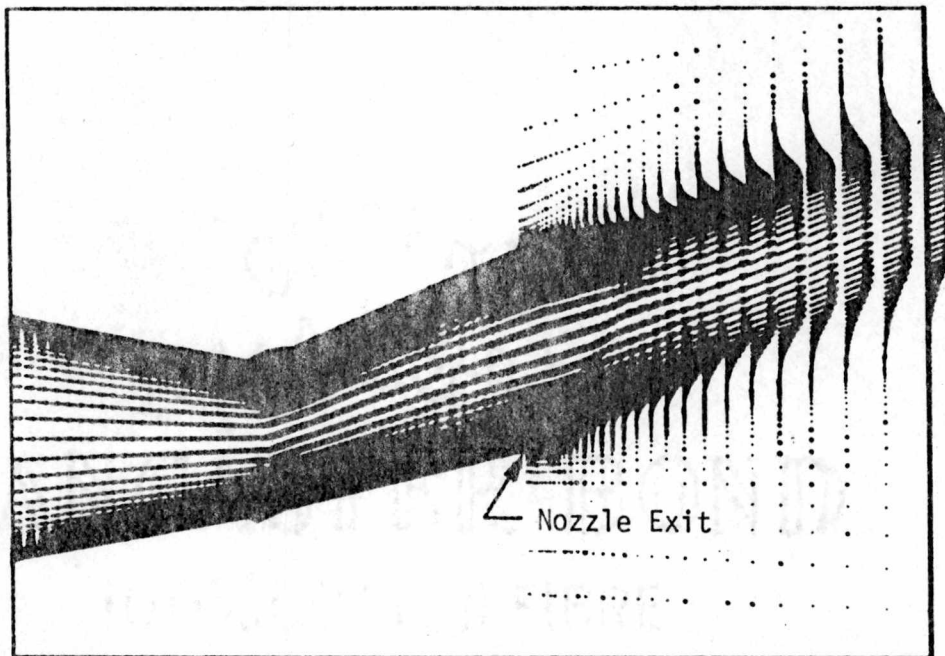


Figure 14. 2D solution (NPR = 3.0) velocity vector plot (top) and isomach plot $M = 1.0$ (bottom).

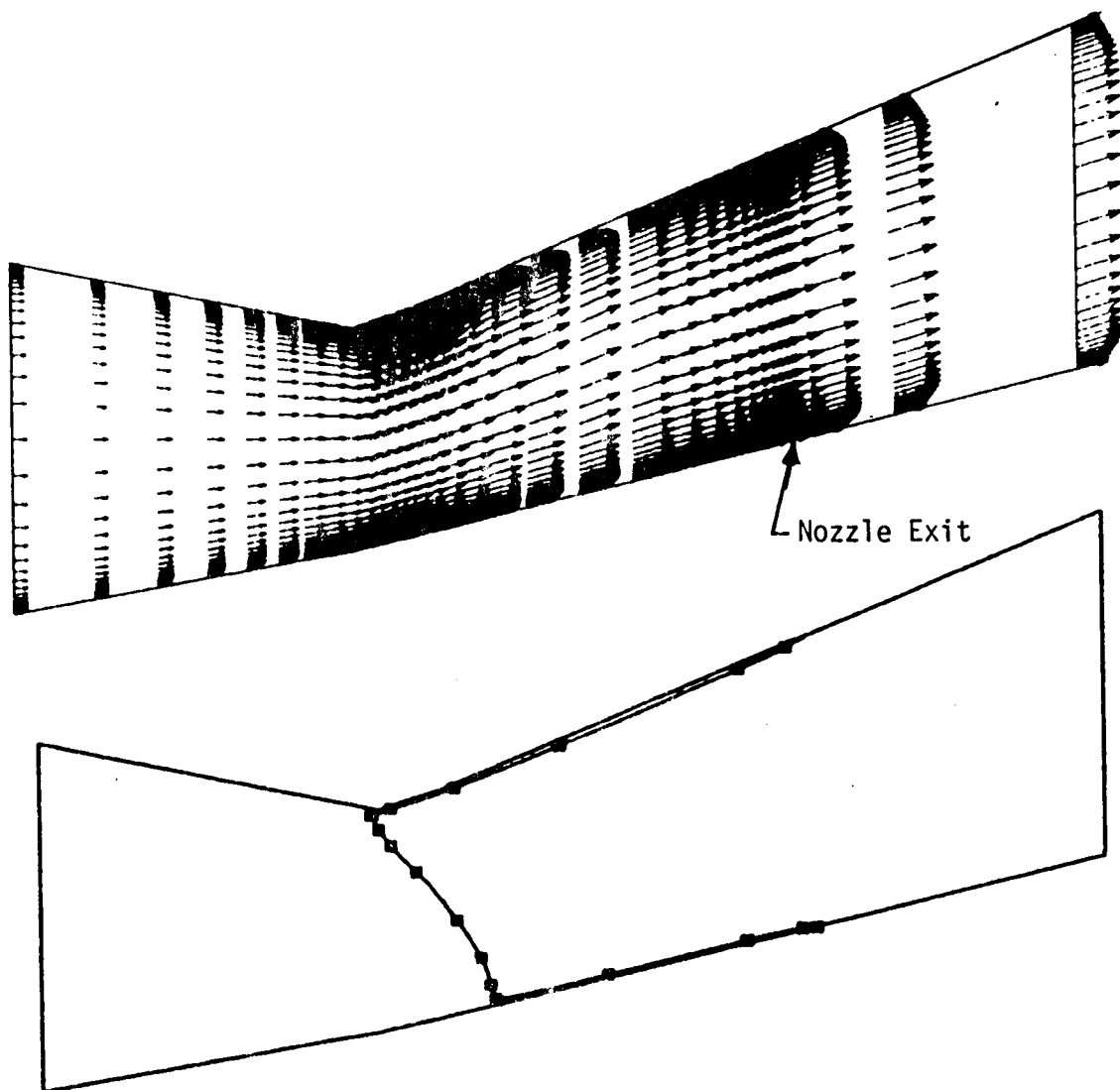


Figure 15. 3D solution (NPR = 3.0) velocity vector plot (top) and isomach plot, $M = 1.0$ (bottom).

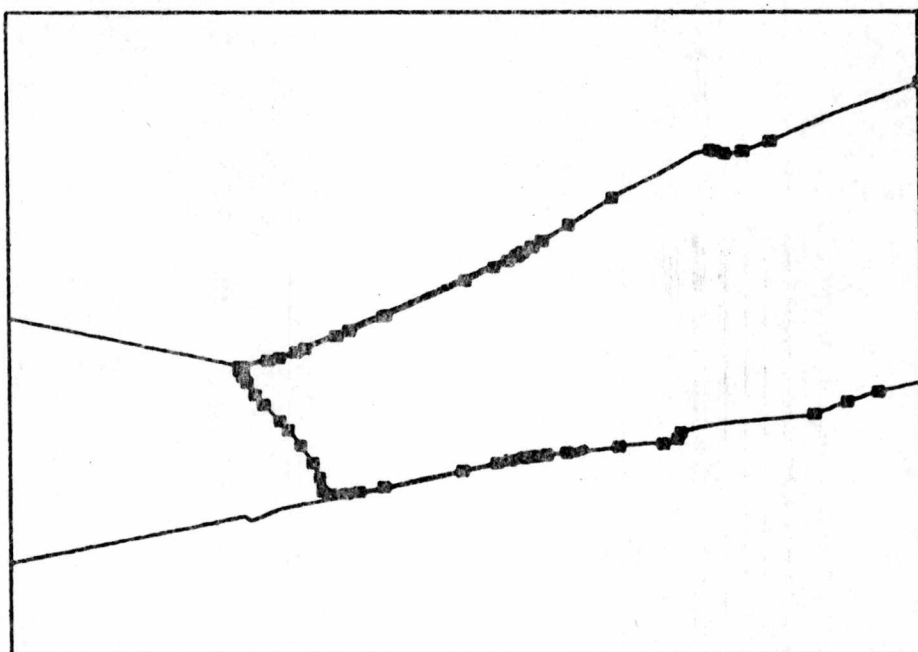
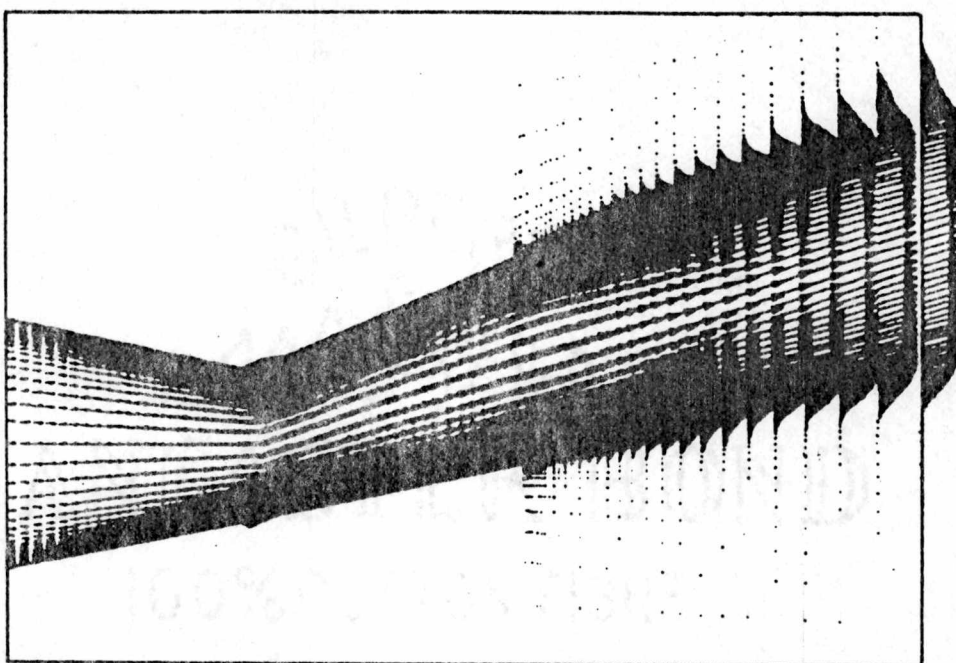


Figure 16. 2D solution (NPR = 6.3) velocity vector plot (top) and isomach plot, $M = 1.0$ (bottom).

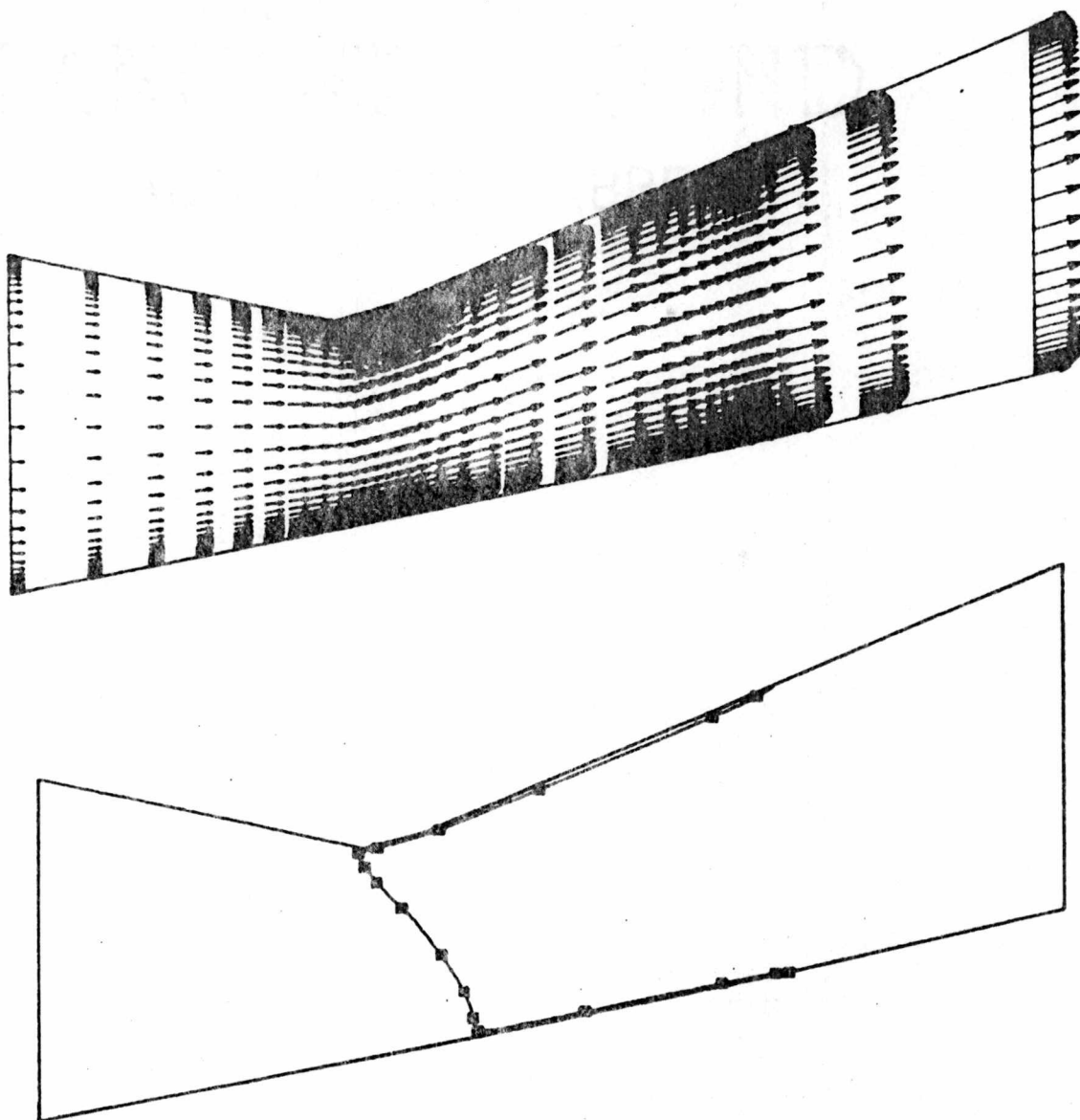


Figure 17. 3D solution ($NPR = 6.3$) velocity vector plot (top) and isomach plot, $M = 1.0$ (bottom).

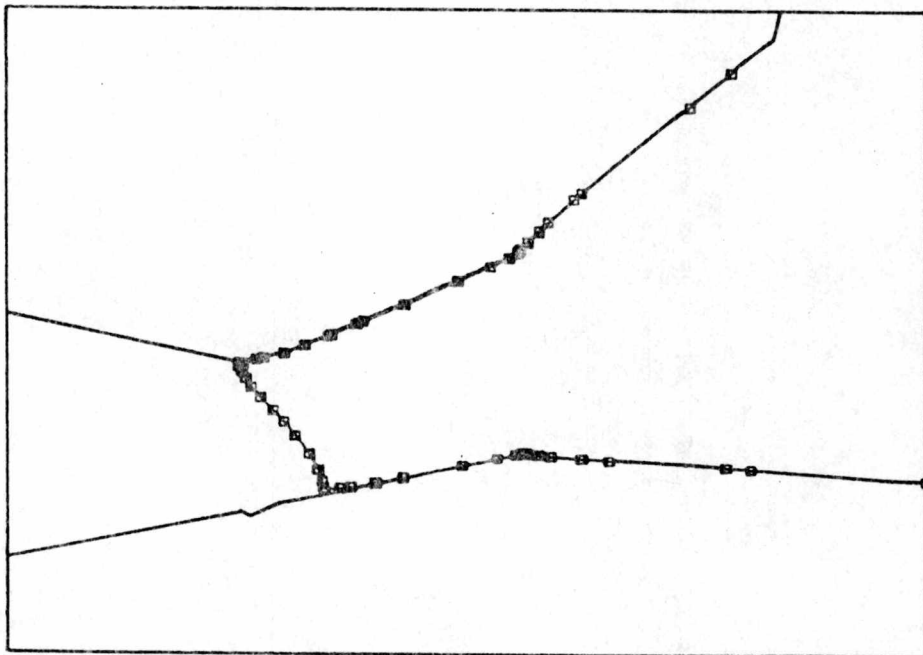
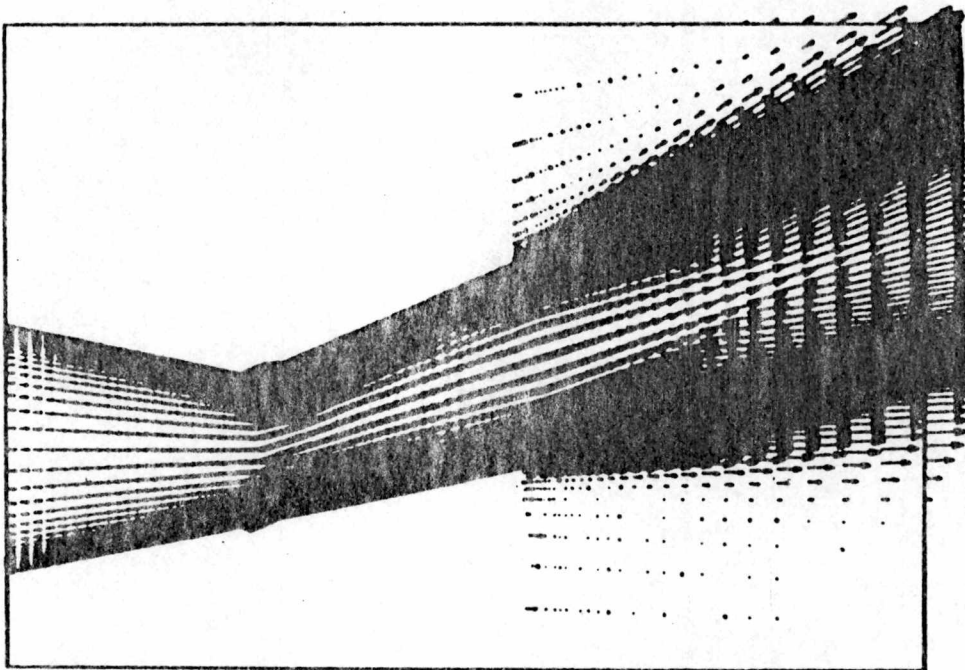


Figure 18. 2D solution ($NPR = 12.1$) velocity plot (top) and isomach plot, $M = 1.0$ (bottom).

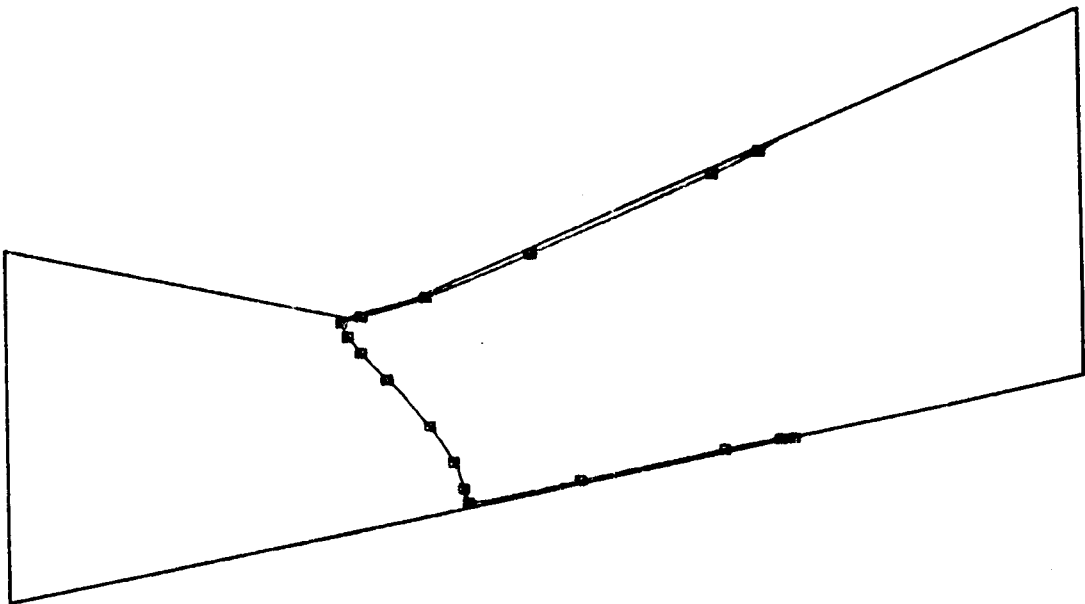
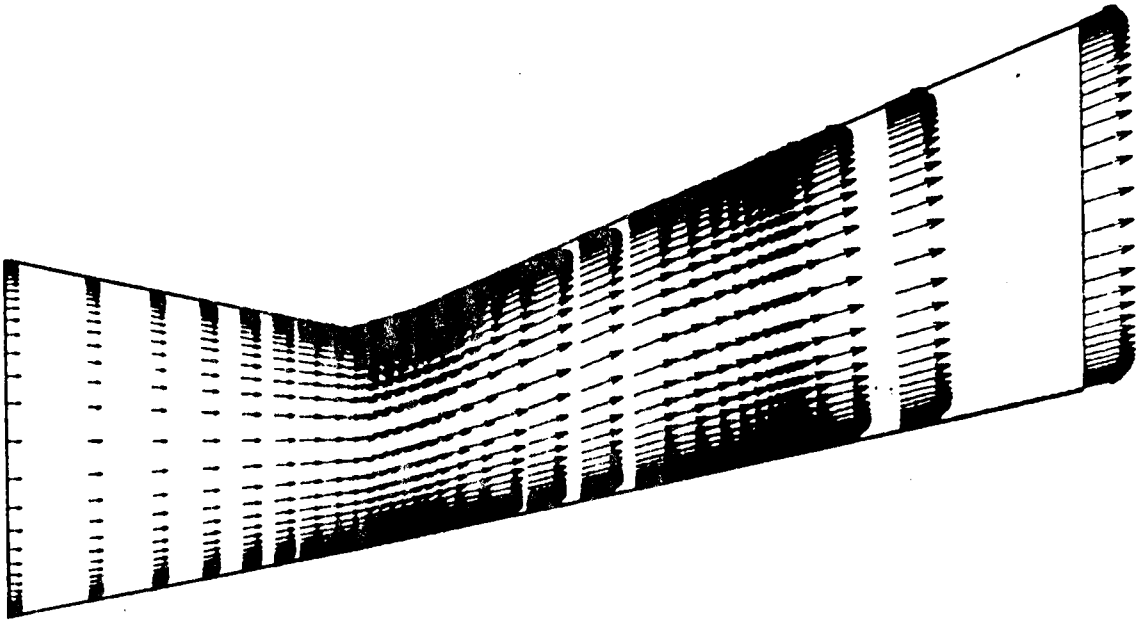


Figure 19. 3D solution (NPR = 12.1) velocity vector plot (top) and isomach plot, $M = 1.0$ (bottom).

pressure at the nozzle exit is conservatively 10 percent higher than ambient pressure) is shown in Figure 16. A highly under-expanded nozzle flow field is shown in Figure 18. The corresponding 3D solutions depicted in Figures 15, 17, and 19 do not attempt to resolve either of these aforementioned phenomena due to the modeling technique. It should be noted, however, that a high degree of flow-resolution beyond the nozzle exit will not have a significant influence on performance predictions because the flow is supersonic throughout this region. The sonic location at the nozzle throat for the 2D simulations (shown in the lower half of Figures 14, 16, and 18) qualitatively agrees with the corresponding 3D simulation sonic line plots (shown in the lower half of Figures 15, 17, and 19).

VII. NOZZLE ASPECT RATIO PARAMETRIC STUDY

Comparison of the 2D and 3D integrated results indicated less than 1.43 percent difference between the resolved force and effective vector angle. This indicates that for the nozzles modeled with an aspect ratio of 1.3 the nozzle sidewalls did not have a large influence on the flow field simulations, and the 3D code predicted a predominantly 2D flow field.

A parametric study of nozzle aspect ratio (in which the nozzle width was reduced while the nozzle height was held constant) was performed in order to investigate the limits of the more economical 2D CFD simulation results. A second reason for performing the aspect ratio parametric study was to investigate the possibility of more severe disagreement between 2D and 3D solutions to 2D/CD nozzle flow fields for future exhaust nozzles which could perhaps have a aspect ratio less than 1.3. The configuration chosen was the NPR=6.3 case. For the NPR=6.3 case the nozzle was scheduled such that the static pressure at the nozzle exit was closer to that of the ambient pressure than for the other two cases, thus reducing the associated expansion and angularity losses, and more accurately representing the nozzle scheduling used in flight. The aspect ratio was varied by reducing the width of the nozzle (see

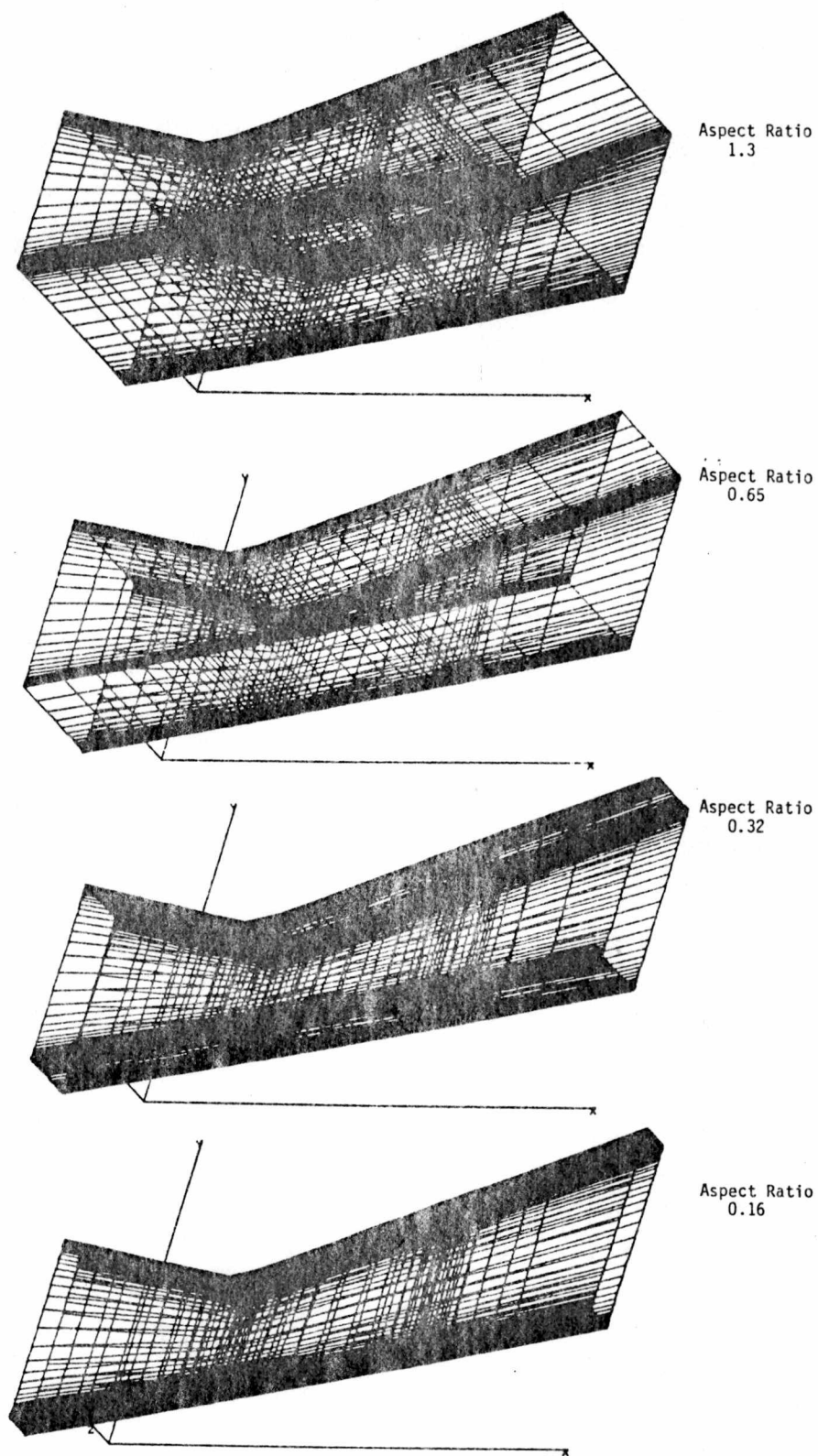


Figure 20. Surfaces of grids used in the aspect ratio parametric study.

Figure 20). The results of the parametric study are shown in Table 3, and Figure 21.

These results indicate that there is less than one percent difference in mass flow, thrust, and effective vector angle between 2D and 3D solutions for nozzle aspect ratios greater than one for this nozzle configuration. The cross flow velocity plots (velocity vector plots projected onto the nozzle exit plane) shown in Figures 22 thru 24 show that the PARC3D code predicted low w-component cross flow velocities for all of the nozzle flow fields simulated. It can be inferred from the cross flow plots that the influence of the sidewall viscous effects on the flow field becomes larger as the aspect ratio is reduced.

Table 3. Aspect ratio parametric study comparison (NPR=6.3)

	2D Solution	3D Solution	$\frac{[2D-3D]}{2D} * 100$
<u>Aspect Ratio = 1.30</u>			
Mass flow	117.6 lbm/s	118.1 lbm/s	-0.43 %
Axial force	14852.0 lbf	14748.0 lbf	0.70 %
Vertical force	4704.0 lbf	4720.0 lbf	-0.34 %
Effective vector	17.6 deg	17.7 deg	-0.56 %
<u>Aspect Ratio = 0.65</u>			
Mass flow	58.8 lbm/s	58.9 lbm/s	-0.17 %
Axial force	7426.0 lbf	7358.7 lbf	0.91 %
Vertical force	2352.0 lbf	2323.7 lbf	1.20 %
Effective vector	17.6 deg	17.53 deg	0.40 %
<u>Aspect Ratio = 0.32</u>			
Mass flow	29.4 lbm/s	29.2 lbm/s	0.68 %
Axial force	3713.0 lbf	3626.8 lbf	2.32 %
Vertical force	1176.0 lbf	1123.4 lbf	4.47 %
Effective vector	17.6 deg	17.2 deg	2.27 %
<u>Aspect Ratio = 0.16</u>			
Mass flow	14.7 lbm/s	14.28 lbm/s	2.86 %
Axial force	1856.5 lbf	1758.1 lbf	5.30 %
Vertical force	588.0 lbf	527.8u lbf	10.23 %
Effective vector	17.6 deg	16.71 deg	5.10 %

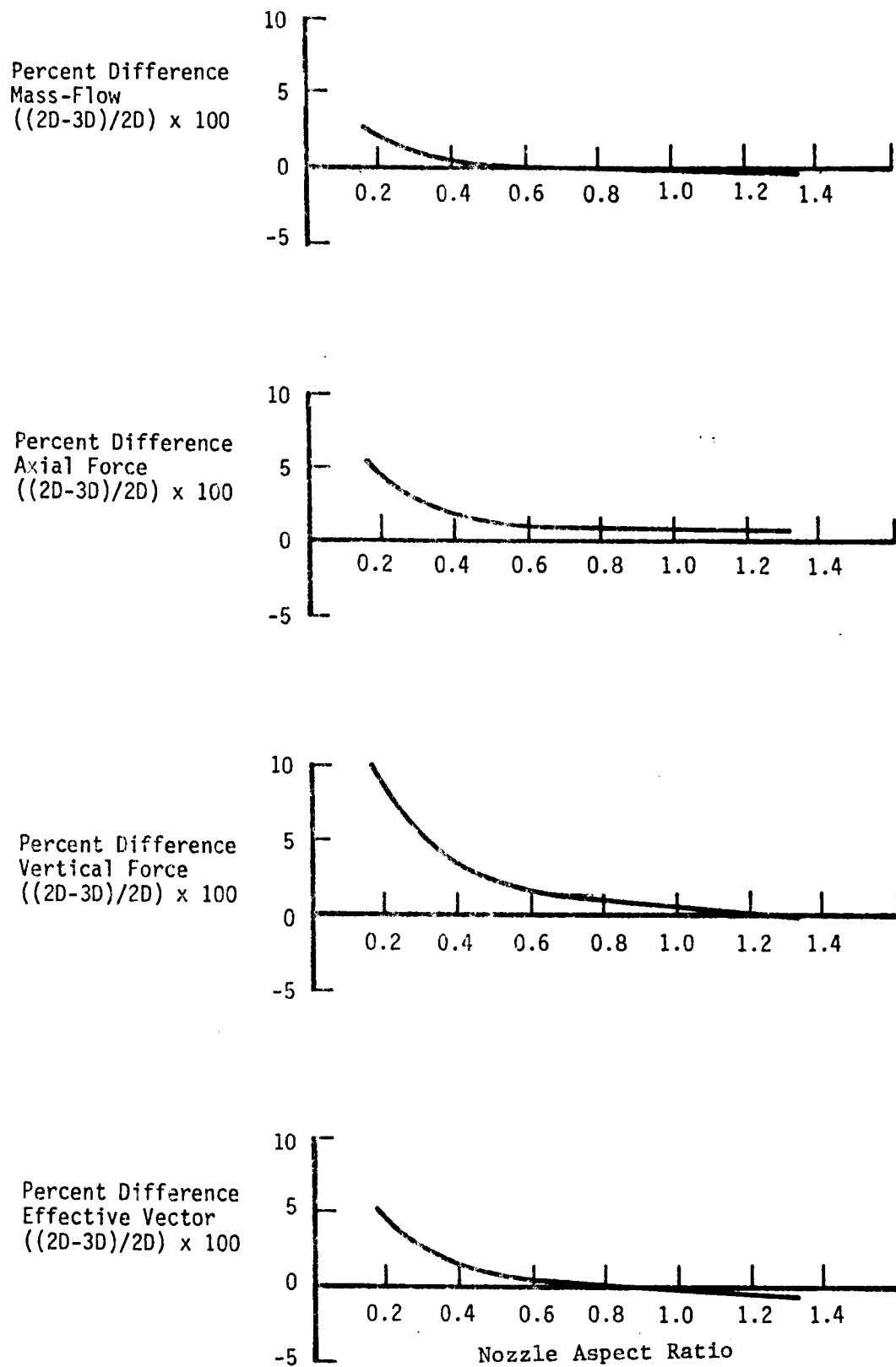
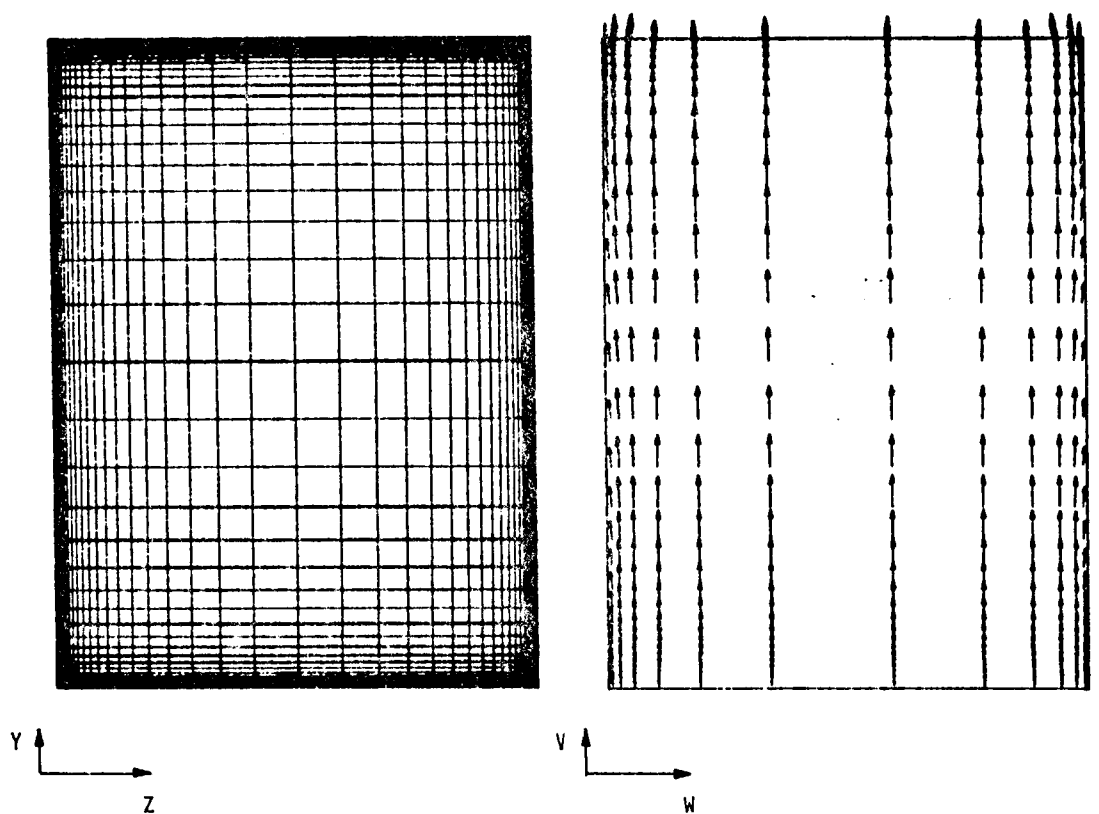


Figure 21. Percent difference curves for mass-flow, axial force, vertical force, and effective vector angle.



a. Nozzle exit grid

b. Cross flow velocity vectors

Figure 22. Cross flow plots
(aspect ratio = 0.65).

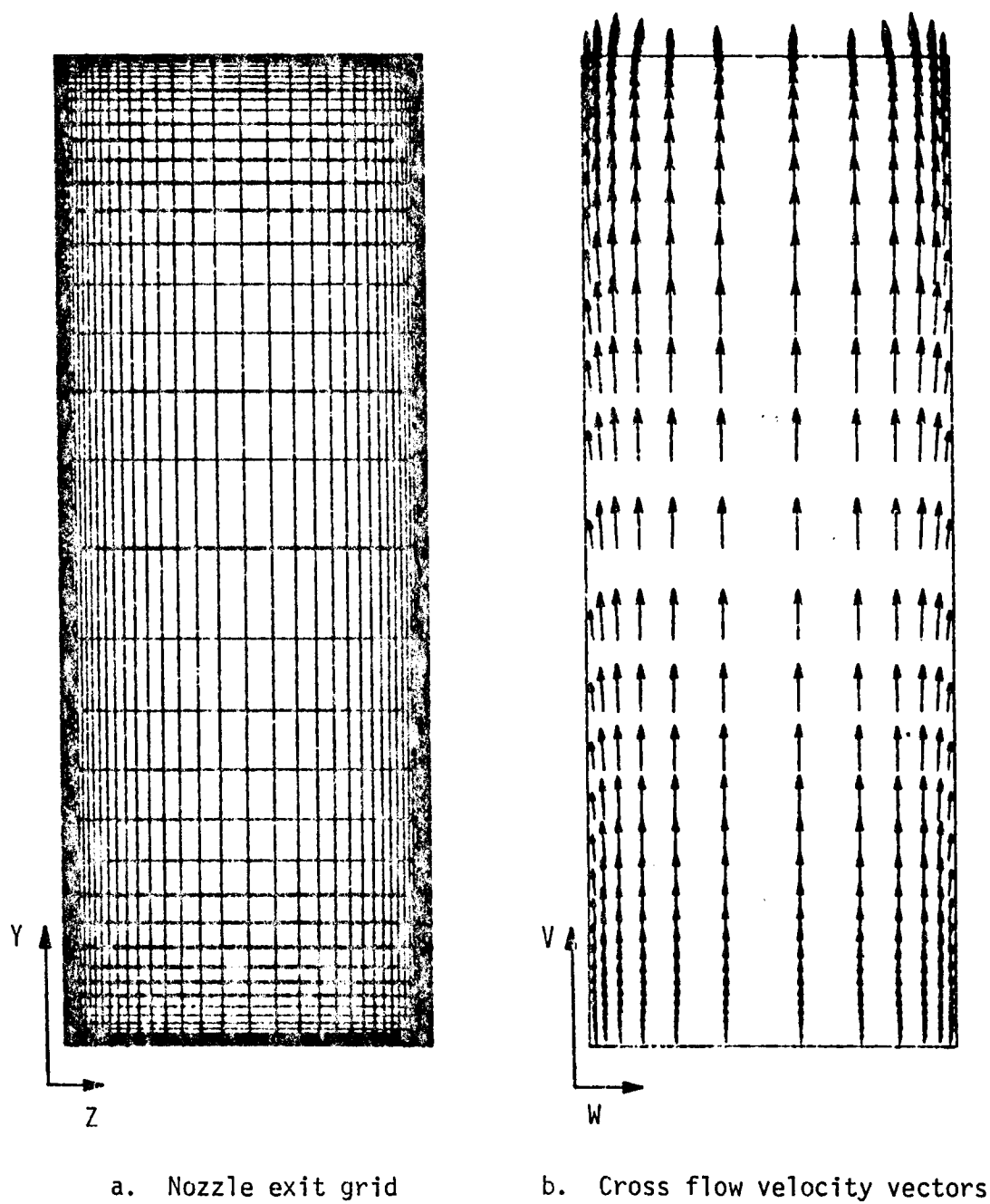


Figure 23. Cross flow plots
(aspect ratio = 0.32).

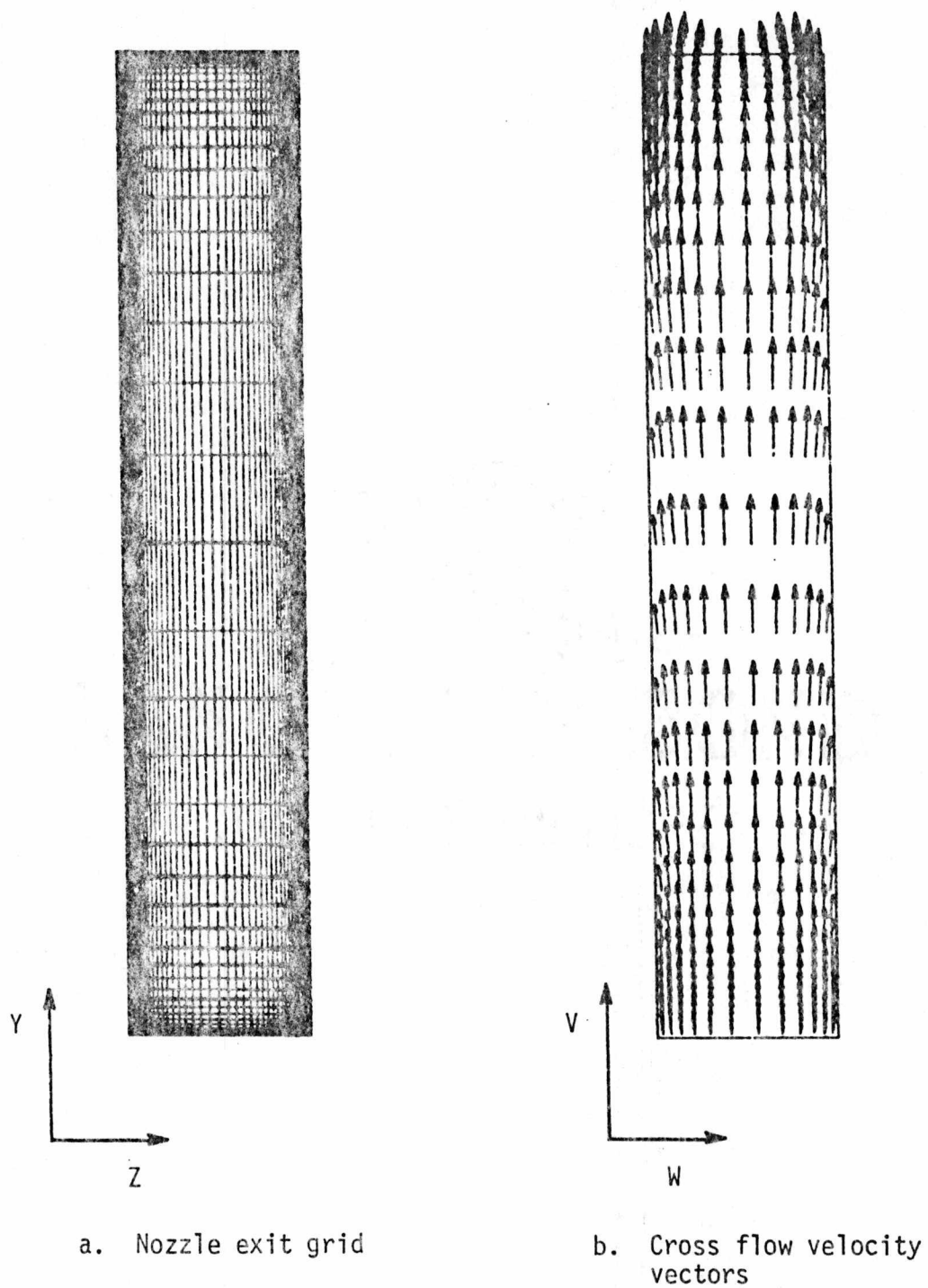
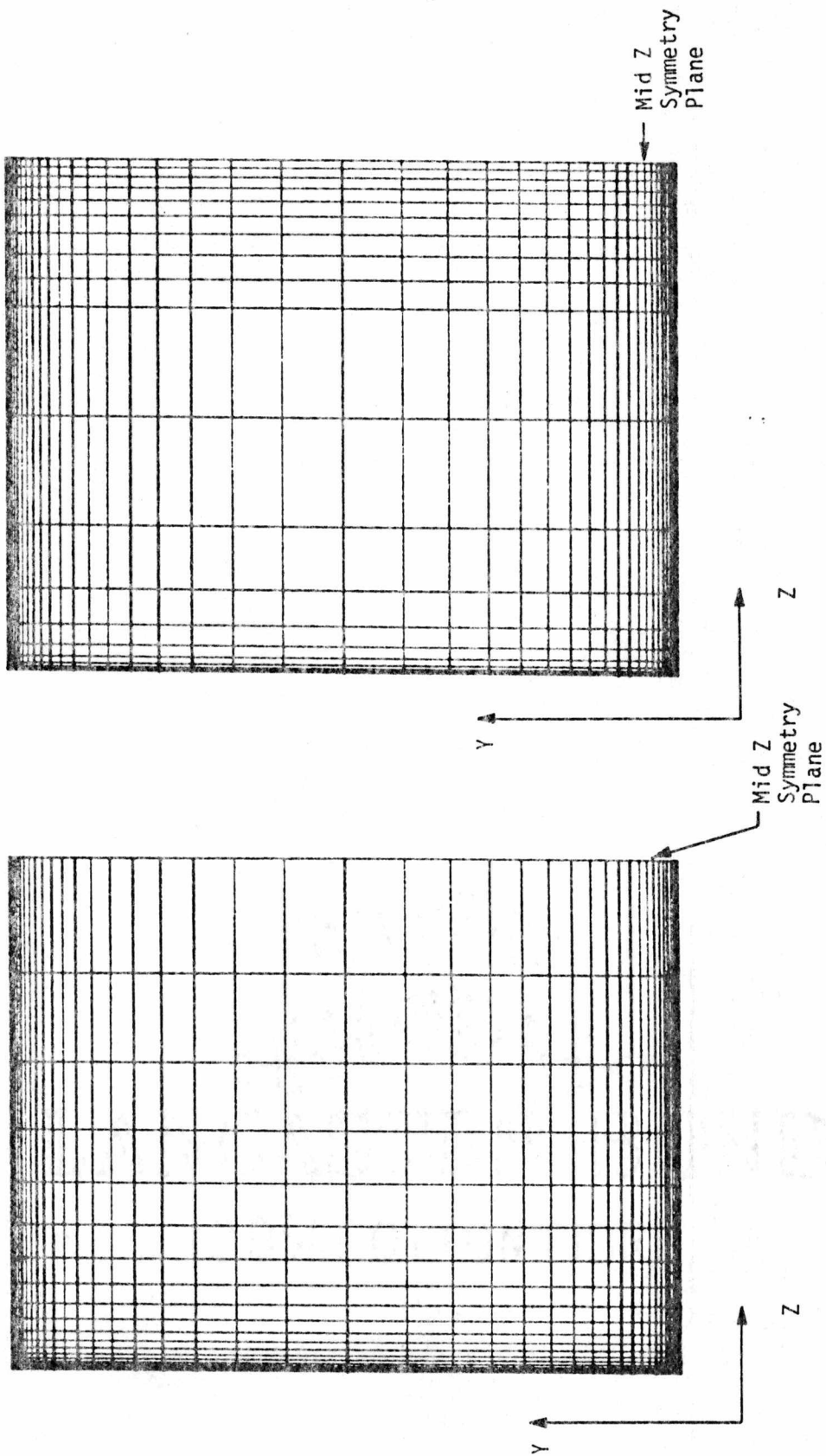


Figure 24. Cross flow plots
(aspect ratio = 0.16).

VIII. SYMMETRY PLANE INVESTIGATION

In order to reduce the number of grid points required to perform a 3D CFD simulation it is often possible to re-scope the 3D problem. For example, for the geometry currently being considered (the NPR=6.3 case) the mid-z plane is a geometric plane of symmetry. For many flow applications, this plane can also be assumed a flow plane of symmetry across which no mass passes and on opposite sides of which the flow is mirror imaged. Such planes of flow symmetry are important in complex 3D applications since, in principal at least, one needs only to solve half the problem and infer the solution in the other half with corresponding savings in computational resources and labor. A second plane of symmetry, common in many 3D applications along the mid-y plane, is destroyed once the nozzle is vectored as currently configured. This vectored mode also prohibits reciprocating the aspect ratio. Figure 25 shows the grid spacing (at the nozzle exit plane) that was used in the symmetry plane study.

Table 4 shows performance results for the NPR = 6.3 case. The first column exhibits results using a full 3D grid (no plane of symmetry assumed). The second column gives results in which a mid-plane of symmetry was assumed, but in which the grid was otherwise unchanged. Finally the third



a. Grid loosely packed at symmetry plane b. Grid tightly packed at symmetry plane

Figure 25. Symmetry plane investigation.

Table 4. Symmetry plane investigation (NPR=6.3)

Comparison of Solutions Capitalizing on Symmetry Plane			
	3D 35x40x40	3D 35x40x20*	3D 35x40x20#
Mass flow, lbm/s	118.1	111.0	117.6
Axial force, lbf	14748.0	13883.0	14724.0
Vertical force, lbf	4720.0	4400.0	4694.0
Effective vector	17.7	17.6	17.7

*Grid not packed against the slip wall plane of symmetry.

#Grid repacked against the slip wall plane of symmetry.

column corresponds to a simulation in which the plane of symmetry is again assumed, but in which the grid points are packed in the neighborhood of the symmetry plane inviscid boundary in order to avoid a loss of accuracy in applying the boundary conditions. A review of Table 4 shows significant effects of grid spacing on computed mass flow, and axial and vertical force. The symmetry plane boundary condition used here is actually the same boundary condition as the slip wall boundary condition previously discussed. The flow properties at the wall are determined by extrapolation of density, pressure, and velocity magnitude from those points adjacent to the slip wall. In the case where the grid is loosely packed near the wall errors induced by extrapolating over a large distance are introduced into the computation. For the case where the grid is tightly packed the influence of the errors introduced by extrapolation is reduced.

IX. CONCLUSIONS

A comparison has been made between solutions generated using the PARC2D and PARC3D codes. The parameters that were compared were mass flow, axial gross thrust, and vertical gross thrust. The simulations have been performed at three nozzle pressure ratios, with uniform nozzle inlet boundary conditions. Based on the results presented herein, it has been determined that:

- 1) Given the assumption of uniform total pressure and total temperature profiles at the nozzle inlet plane, the simulated flow field in 2D/CD nozzles with aspect ratios greater than 1 can be modeled using the PARC2D code with significantly fewer computer resources than required for the PARC3D code, while providing results for mass flow, axial gross thrust, and vertical gross thrust within 2 percent of the PARC3D results.

- 2) For the cases studied the PARC2D code was found to require approximately 74 percent less computer storage and 93 percent less CRAY-XMP CPU time than required by the PARC3D code in order to provide an equally converged flow field solution.

- 3) For the 2D/CD nozzles simulations performed (for nozzles with aspect ratios larger than 0.65) the difference between the effective vector angles predicted with the

PARC2D code and the PARC3D code were within 0.62 percent, indicating that 2D flow field simulations are useful for the determination of effective vector angle.

4) The 2D model has limited use for resolved gross thrust performance predictions. The resolved gross thrust predictions calculated using the two-dimensional code and the three-dimensional code were only found to agree within 1.5 percent for the simulations performed for a nozzle with an aspect ratio of 1.3.

5) The disagreement between the two-dimensional results and three-dimensional results for mass flow, resolved gross thrust, and effective vector angle increase rapidly as the nozzle aspect ratio is reduced below 1.

6) It is important to pack grids near a plane of symmetry in order to reduce errors introduced by extrapolation.

X. RECOMMENDED FUTURE EFFORTS

The CFD simulation techniques that are used for axisymmetric nozzle internal performance predictions are well established; however, this is not true for advanced 2D/CD nozzles. The recommended future efforts should be aimed at developing and calibrating CFD simulation software for this application. A possibility of increasing the value of the less expensive PARC2D simulations arises from the development of an experience base from which the 3D flow effects can be quantified and used to adjust 2D computational results.

Flow Field Measurements

The CFD simulations presented in this thesis did not use actual flow field measurements because they were not available. It is recommended that future CFD simulations incorporate detailed flow field measurements. This should be done in order to provide the best boundary conditions possible for the CFD solutions and in order to provide experimental measurements to further validate the PARC CFD simulation software.

Quantify Secondary Flow Effects

The CFD simulations presented in this thesis did not predict high degrees of secondary flow (mixing and cross-flows). This is believed to be in part based on the assumed uniform inflow boundary conditions. The influence of secondary flow field effects in 2D/CD nozzles must be quantified. If the influence of secondary effects on nozzle performance are determined to be small (less than 0.5 percent), then it may not be necessary to model them with a detailed 3D CFD simulation especially if it is possible to make empirical adjustments accurately for swirl and mixing to the 2D solutions.

Transition Duct Modeling

The CFD simulations presented in this thesis did not include the transition zone (the portion of the tailpipe that allows a circular tailpipe to connect to a rectangular nozzle via a super ellipse). Experimental and CFD evidence exists that shows circular to rectangular cross section duct flows induce a vortex pair which propagates into the rectangular section of the nozzle Ref. [13]. It is recommended that the PARC code be used to model transition zones for which experimental data exist in order to

determine the adequacy of the PARC code for modeling these flows and to determine the influence of the transition zone on nozzle performance.

Leakage and Cooling Flow Modeling

The modeling of leakage and cooling flow rates and distributions is another area which was not addressed in this thesis. A determination must be made concerning whether or not it is practical to model leakage and cooling flow rates using CFD. It should be determined whether or not leakage and cooling flow rates can be accounted for based on rig test data, manufacturers predictions or measurements during tests.

Nozzle Wall Heat Transfer

The CFD solutions presented in this thesis utilized adiabatic nozzle wall boundary conditions. It is recommended that more realistic wall temperature distributions be incorporated into the calculations in order to evaluate the effect of nozzle wall heat transfer on nozzle performance predictions.

Real Gas Effects

The CFD simulations presented in this thesis also did not model real gas effects. The influence of real gas effects needs to be quantified and either modeled or empirically accounted for.

LIST OF REFERENCES

LIST OF REFERENCES

1. Capone, F. J. "The Nonaxisymmetric Nozzle-It is for Real," AIAA Paper No. 79-1810, August 1979.
2. Dusa D. J., Speir D. W., and Dunbar D. K. "Multi-Functional Nozzles For Advanced Weapon Systems" SAE Paper No. 831426, Aerospace Congress and Exposition, Long Beach, California, October 3-6, 1983.
3. McLafferty G. H. and Peterson, J. L. "Results of Test of Rectangular Vectoring/Reversing Nozzle on an F100 Engine," AIAA Paper No. 83-1285, June, 1983.
4. Stevens, H. L. Thayer, E. B. , and Fullerton, J. F. "Development of the Multi-function 2D/CD Nozzle" AIAA Paper No. 81-1491, July 1979.
5. Berrier B. L. and Mason M. L., "Static Investigation of Post-Exit Vanes for Multi-axis Thrust Vectoring" AIAA Paper No. 87-1834, AIAA 23rd Joint Propulsion Conference, San Diego, California. June 29-July 2, 1987.
6. Reddy, D. R. and Harloff, G. J. "Three-dimensional Viscous Flow Computations of High Area Ratio Nozzles for Hypersonic Propulsion" AIAA Paper No. 88-0474, AIAA 26th Aerospace Sciences Meeting, Reno, Nevada, January 11-14, 1988.
7. Cooper, G. K., Jordan, J. L., and Phares W. J. "Analysis Tool For Application to Ground Testing of Highly Underexpanded Nozzles." AIAA Paper No. 87-2015, AIAA/SAE/ASME/ASEE 23rd Joint Propulsion Conference, June 1987.
8. Mourey W. L. " Test Data Report - Flow Characteristics Including Sonic Line Determination For Four Convergent Nozzles" Boeing Company Report TR-5478-1 April 8, 1968.
9. Oates, G. C. "The Aerothermodynamics of Aircraft Gas Turbine Engines," AFAPL-TR-78-52, July, 1978.
10. Zucker, Robert D. "Fundamentals of Gas Dynamics," Matrix Publishers, Inc., 1977.

11. Berrier, B. L. and Re, R. J. "Effect of Several Geometric Parameters on the Static Internal Performance of Three Nonaxisymmetric Nozzle Concepts." NASA TP-1468, 1979.
12. Thomson, J. F. Numerical Grid Generation Elsevier Science Publishing Co., Inc. New York, New York, 1982.
13. Patrick W. P. and McCormick D. C. "Circular to Rectangular Duct Flows A Benchmark Experimental Study" SAE Technical Paper Series 871776, Aerospace Technology Conference and Exposition, Long Beach, CA October 5-8, 1987.
14. Pulliam, T. H. and Steger, J. L. "Implicit Finite-Difference Simulations of Three-Dimensional Compressible Flow." AIAA Journal, Vol. 18, No. 2, February 1980, pp. 159-167.
15. Pulliam, T. H. and Steger, J. L. "Euler and Thin Layer Navier-Stokes Codes: ARC2D, ARC3D." Computational Fluid Dynamics- A Workshop held at the University of Tennessee Space Institute, Tullahoma, Tennessee, (UTSI Publication E02-4005-023-84), March 12-16, 1984. pp. 15.1-15.85.
16. Anderson, D.A. Tannehill, J.C., and Pletcher, T. H. Computational Fluid Mechanics and Heat Transfer, Hemisphere Publishing Corporation, Washington, D. C., 1984.
17. Beam, R. and Warming, R. F. "An implicit Finite-Difference Algorithm for Hyperbolic Systems in Conservation-Law form." Journal of Computational Physics, Vol. 22, No. 1 September 1976, pp.87-110.
18. Jameson, A., Schmidt, W. and Turkle, E. "Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes." AIAA Paper No. 81-1259, AIAA 14th Fluid and Plasma Dynamics Conference, Palo Alto, California, 1981.
19. Cooper, G. K., "Towards a General Purpose Navier-Stokes Code," Abstracts, First World Congress on Computational Mechanics, Vol. I, Austin, Texas, September 1986.

20. Baldwin B. H. and Lomax H., "Thin layer Approximation and Algebraic Model For Separated Turbulent Flows" AIAA Paper No. 78-257, AIAA 16th Aerospace Sciences Meeting. Huntsville, Alabama, January 16-18, 1978.
21. Smith, R. E., Jr., and Wehofer, S. "Measurement of Engine Thrust in Altitude Ground Test Facilities." AIAA Paper No. 82-0572, AIAA 12th Aerodynamic Testing Conference, Williamsburg, Virginia, March 1982.
22. Ames Research Staff. "Equations, Tables, and Charts for Compressible Flow." NACA Report 1135, 1953.
23. Strait, D. M. and Cullom, R. R., "Thrust Performance of a 2D/CD Nonaxisymmetric Exhaust Nozzle on a Turbojet Engine at Altitude", NASA Technical Paper 2171, June 1983.
24. Shapiro, A. H.: The Dynamics and Thermodynamics of Compressible Fluid Flow. Vol. I. Ronald Press Co., 1953.
25. Burden, R. L., Faires, J. D., and Reynolds, A. C., Numerical Analysis, Prindle, Weber and Schmidt, Boston, Mass., 1978.
26. MacCormack, R. W. "Current Status of Numerical Solutions of the Navier-Stokes Equations," AIAA-85-0032, Jan. 1985.
27. Abramovich, G. N., The Theory of Turbulent Jets, MIT Press, Cambridge, Massachusetts, 1963.
28. Sobota T. H. and Marble F. E. "An Experimental and Numerical Investigation of Swirling Flows in a Rectangular Nozzle," AIAA Paper No. 87-2108, June, 1987.
29. Sova G. J. "Application of Navier-Stokes Analysis to Predict the Internal Performance of Thrust Vectoring Two-Dimensional Convergent-Divergent Nozzles" AIAA Paper No. 88-2586, June, 1988.
30. Re and Leavitt, L. D., "Static Internal Performance Including Thrust Vectoring and Reversing of Two-Dimensional Convergent-Divergent Nozzles," NASA TP-2253, February 1984.

APPENDIX

PARC CODE THEORY

The purpose of this appendix is to cite the fundamental equations and present the fundamental approach used by the PARC codes to solve the conservation equations. Derivation of the conservation equations of fluid mechanics and details concerning the implementation of the algorithm used to solve these equations have been well documented elsewhere, Refs. [14, 15, and 16].

The PARC codes are flow field simulation programs that calculate the thermodynamic and kinematic properties of a fluid flow at discrete points within the flow, based on specified boundary geometry and appropriate flow conditions on these boundaries. The basis of the algorithms used in the PARC codes is the complete Navier-Stokes equations in conservation law form. The viscous terms can be selectively calculated so that a thin-layer simulation can be performed or an inviscid (Euler) flow field calculated.

The governing differential equations used to model the 2D/CD nozzle flow fields are the Reynolds-averaged Navier-Stokes equations. Assumptions basic to the form of the Navier-Stokes equations used were that of :

single species, single phase, electrically neutral, inertial reference frame, no sources or sinks, and no body forces. Basic to the auxiliary relations (e.g. equations of state) are the following assumptions: the assumption of a Newtonian fluid obeying the Fourier heat conduction law; the assumption of a thermally and calorically perfect fluid; the assumption of constant Prandtl number; the use of Sutherland's viscosity law and Stokes hypothesis.

For notational convenience, use is made of the Kronecker delta:

$$\delta_{ij} = \begin{cases} 1, & \text{if } i=j \\ 0, & \text{if } i \neq j \end{cases} \quad (\text{A1})$$

and of the Einstein summation convention:

$$u_k u_k = \sum_{j=1}^N u_j u_j \quad (\text{A2})$$

(where repeated indicies in a product indicate summation over the range of the indicies). The indicies range from 1 to 2 for the PARC2D version of the code, and from 1 to 3 for the PARC3D version of the code. Using this convention the system of partial differential equations (continuity, momentum, and energy equations) can be expressed in the nondimensional Cartesian conservation law form:

$$\frac{\partial Q}{\partial t} + \frac{\partial F_j}{\partial x_j} = \frac{1}{R_e} \frac{\partial G_j}{\partial x_j} \quad (A3)$$

Where Q is a vector containing the conservation variables:

$$Q = \begin{bmatrix} \rho \\ \rho u_i \\ E \end{bmatrix} \quad (A4)$$

the F_j vectors represent the inviscid flux vectors:

$$F_j = \begin{bmatrix} \rho u_j \\ \rho u_i u_j + P \delta_{ij} \\ (E + P) u_j \end{bmatrix} \quad (A5)$$

and the G_j vectors represent the viscous flux vectors:

$$G_j = \begin{bmatrix} 0 \\ \tau_{ij} \\ u_k \tau_{ij} - q_j \end{bmatrix} \quad (A6)$$

The viscous stress tensor takes the following form for a Newtonian fluid:

$$\tau_{ij} = \mu \left[\frac{u_i}{x_j} + \frac{u_j}{x_i} \right] + \lambda \frac{u_k}{x_k} \delta_{ij} \quad (A7)$$

Assuming a Fourier heat conduction law, the heat flux vector is:

$$q_j = - \frac{\kappa}{\beta_r P_r} \frac{\partial T}{\partial x_j} \quad (A8)$$

E, the total energy per unit volume is defined to be:

$$E = \rho \left[e + \frac{1}{2} u_k u_k \right] \quad (A9)$$

Definitions of fluid property notation and the associated non-dimensionalizing parameters are listed below:

PROPERTY	NOTATION	NON-DIMENSIONALIZING PARAMETER
Density	ρ	ρ_r
Pressure	P	$\rho_r a_r^2$
Temperature	T	T_r
Internal energy per unit mass	e	a_r^2
First Coefficient of viscosity	μ	μ_r
Second Coefficient of viscosity	λ	μ_r
Thermal conductivity	K	K_r
Velocity components	u_i	a_r
Cartesian coordinates	x_i	x_r
Time	t	x_r/a_r

The "r" subscript indicates an arbitrary reference fluid state, and "a_r" is the corresponding reference sound speed. The following dimensionless parameters are also required:

$$R_e = \rho_r a_r x_r / \mu_r \quad (A10)$$

$$P_r = c_{pr} \mu_r / K_r \quad (A11)$$

$$\beta_r = a_r^2 / c_{pr} T_r \quad (A12)$$

Where c_{pr} = reference specific heat at constant pressure.

The form of the second coefficient of artificial viscosity used was based on Stokes hypothesis.

$$\lambda = -\frac{2}{3} \mu \quad (A13)$$

From the state postulate of thermodynamics for a simple compressible fluid, the thermodynamic properties can be expressed as functions of density and internal energy.

$$\begin{aligned} P &= P(\rho, e) & \mu &= \mu(\rho, e) \\ T &= T(\rho, e) & K &= K(\rho, e) \end{aligned}$$

However, the versions of the PARC2D and PARC3D code used for this study are based on the assumption of a thermally and calorically perfect gas.

The following nondimensional equations of state are used:

$$P = \rho T / \gamma \quad (A14)$$

$$e = T / \gamma (\gamma - 1) \quad (A15)$$

This allows the pressure and temperature to be expressed as functions of the conservation variables:

$$P = (\gamma - 1) (E - 1/2 u_k u_k) \quad (A16)$$

$$T = \gamma (\gamma - 1) (E / \rho - 1/2 u_k u_k) \quad (A17)$$

Here the ratio of specific heats, γ , is a constant and $\beta_r = \gamma - 1$.

The perfect gas version of the PARC codes also use the assumption of constant Prandtl number and the Sutherland viscosity law:

$$\kappa = \mu \tag{A18}$$

$$\mu = T^{3/2}(1+T_S)/(T+T_S) \tag{A19}$$

where T_S is the nondimensional Sutherland temperature.

In order to provide easy discretation of the Navier-Stokes equations, they are re-expressed in terms of general curvilinear coordinates while retaining the strong conservation law form.

The Beam and Warming approximate factorization algorithm , Ref. [17], forms the heart of the PARC code. This algorithm is an implicit scheme that solves the set of equations produced by central differencing the Navier-Stokes equations on a regular grid. Since these equations are formulated in the strong conservation form for a curvilinear set of coordinates, the resulting algorithm is quite general with the desirable features of global conservation and shock capturing. Its alternating direction implicit style formulation makes this implicit scheme economical in comparison to other

implicit and explicit formulations. Use of the Beam and Warming algorithm in a Navier Stokes code has been reported by Pulliam and Steger Ref. [14]. The original Pulliam and Steger programs, known as the AIR2D and AIR3D, were updated by Pulliam [10] who diagonalized the implicit matrices for more efficient execution times. The diagonalized versions have been shown to significantly improve the efficiency of the block matrix inversions by replacing the inversion of a system of block tridiagonal equations (pentadiagonal if implicit artificial dissipation is included) with the inversion of a set of scalar tridiagonal (pentadiagonal) equations. Pulliam also modified the artificial dissipation so that it was treated completely implicitly and included a Jameson-style, Ref. [18], second-order term for improved shock capturing. These codes, termed the ARC2D and ARC3D program, have been in widespread use within the aerodynamics community.

The PARC codes, which were developed at AEDC by Cooper Ref. [19], are extensions of the Pulliam-Steger ARC codes. Enhancements made by Cooper in the development of the ideal gas versions of the PARC codes include:

- 1) Addition of code logic to allow specification of boundary-condition type and location arbitrarily

throughout the computational domain strictly through input rather than through code modifications,

2) Acceleration of convergence rate through the addition of an option to allow automatic variation of the time step both spatially and temporally,

3) More generalized turbulence modeling via modifications of the Baldwin-Lomax algebraic turbulence model Ref. [20] in wake mixing regions.

VITA

Matthew Brian Prufert was born in Norman, Oklahoma, on October 5, 1962. He attended elementary schools in California, Iowa, Nebraska, and Tennessee, and was graduated from Germantown High School, in Germantown, Tennessee, in May 1980. The following August he entered Christian Brothers College in Memphis, Tennessee, and in May 1984 he received a Bachelor of Science degree in Mechanical Engineering.

After graduation he began employment with Sverdrup Technology, Inc., at Arnold Engineering Development Center near Tullahoma.

He began taking graduate courses in September 1984 at the University of Tennessee Space Institute, and Received a Master of Science degree in Mechanical Engineering in December of 1988.