MARSHALL GRANT
IN-34-CR
114227
P. 34

LOW THRUST VISCOUS NOZZLE FLOW FIELDS PREDICTION

GRANT NAG8-064

(FIRST YEAR REPORT)
1986-1987

SUBMITTED TO

NATIONAL AERONAUTICS AND SPACE ADMINISTRATION

MARSHALL SPACE FLIGHT CENTER

(NASA-CR-182334) LOW THRUST VISCOUS NOZZLE FLOW FIELDS PREDICTION Annual Report No. 1, 1986-1987 (Alabama A & M Univ.) 34 p Avail: NTIS HC A03/MF A01 CSCL 20D
N88-13551
Unclas
G3/34 0114227

BY

GOANG-SHIN LIAW

SCHOOL OF ENGINEERING & TECHNOLOGY

ALABAMA AGRICULTURAL AND MECHANICAL UNIVERSITY

DECEMBER 1987

SUMMARY

The present work used an existing Navier-Stokes code (PARC2D) to compute the nozzle flow field. Grids were generated by the interactive grid generator codes TBGG and GENIE. All computations were made on the NASA/MSFC CRAY X-MP computer. Comparisons were made between the computations and MSFC in-house wall pressure measurements for CO_2 flow through a conical nozzle having an area ratio of 40. Satisfactory agreements exist between the computations and measurements for different stagnation pressures of 29.4, 14.7 and 7.4 psia, at stagnation temperature of 1060 ^{0}R. However, agreements did not match precisely near the nozzle exit. Several reasons for the lack of agreement are possible. The computational code assumes a constant gas gamma, whereas the gamma i.e. the specific heat ratio foı CO_2 varied from 1.22 in the plenum chamber to 1.38 at the nozzle exit. The computations also assumes adiabatic and no-slip walls. Both assumptions may not be correct. Finally, it is possible that condensation occurs during the nozzle expansion at the low stagnation pressure. The next phase of the work will incorporate variable gamma and slip wall boundary conditions in the computational code and develope a more accurate computer code.

TABLE OF CONTENTS

1. INTRODUCTION

During the last decade, the requirements for more accurate and economical computation methods for predicting fluid flows in practical engineering problems has become increasingly demanding. Since full scale tests or model tests are often expensive and in many cases inconclusive, it is indispensable to rely on the analysis of computational fluid dynamics. It is also important to have adequate numerical methods and computer codes available to help the interpretation of experimental data and to aid vehicle design. The treatments of fluid flows have advanced tremendously in recent years due to the continuous improvements of advanced computer such as CDC, CYBER and the CRAY.

In space applications the rocket engines for attitude controls and orbital maneuvers are often required to produce a thrust less than one pound force. The requirements of such low thrust engine dictate the use of low chamber pressures, low mass flow rates and small nozzles. It results that the nozzle flow is in the low Reynolds number range with viscous effects dissipated across the whole nozzle. An accurate knowledge of their thrust is required for designing the spacecraft control system. Recently the man-made contaminations to the spacecraft from the nozzle exhaust has become an issue, and accurate solutions for the nozzle/plume are required. The flow in such nozzles possesses strong viscous/inviscid interactions at their exit due to the thick boundary layers. Traditional nozzle design techniques, such as the use of the method of characteristics to calculate the inviscid core and boundary layer theorey to compute the displacement thickness, fail to predict the strength of the viscous/inviscid interaction. Therefore, it is necessary to use a full Navier-Stokes code for this purpose.

Based on these premises, this project was proposed to develop an adequate numerical method of solvidng the full Navier-Stokes equations in three years and to predict the two-dimensional mixed subsonic/supersonic flow characteristics of the low thrust converging/diverging nozzle, which the viscous effects are eminent across the entire nozzle.

The present work used a full Navier-Stokes code (PARC2D) to compute the nozzle flow field. Grids were generated using the interactive grid generator codes TBGG and GENIE. All computations were made on the NASA/MSFC CRAY X-MP computer. Comparisons were made between the computations and wall pressure measurements for CO_2 flow through a conical nozzle having an area ratio of 40. Satisfactory agreements exist between the computations and measurements for different stagnation pressures of 29.4, 14.7 and 7.4 psia, at stagnation temperature of 1060 ^{0}R. However, agreement did not match precisely near the nozzle exit. Several reasons for the lack of agreement are possible. The computational code assumes a constant gas gamma, whereas the gamma i.e. the specific heat ratio for CO_2 varied from 1.22 in the plenum chamber to 1.38 at the nozzle exit. The computations also assumes adiabatic and no-slip walls. Both of which may not be correct. Finally, it is possible that condensation occurs during the nozzle expansion for low stagnation pressure. The next phase of the work will incorporate variable gamma and slip wall boundary conditions in the computational code. Our first year's experience on running this existing code enhances the capability to develope a more accurate and efficient computer code for this type of flow problem.

2. OBJECTIVES

The first objective of this project is to compute the CO_2 nozzle flow fields by using existing computer code (PARC2D) and to compare the results with the experimental data. The second objective is to develop a computer code for solving the compressible full Navier-Stokes equations for low thrust type nozzle flow calculations. The third objective is to include the rarefield gas dynamic in this code by adding the velocity and temperature slip conditions along the nozzle wall.

3. APPROACH

3.1 Experimental Test Results

A series of low thrust nozzle tests were performed by Dr Lynn C. Chou and James A. Carter [1] at NASA/MSFC in 1983. The experiment was on a CO_2 nozzle flow field and its associated plume expanding into a highly rarefied space environment. The pressures along the nozzle wall and pitot pressures out of the nozzle along the center line will be used in this project as the reference data base. Figure 1 shows the nozzle configuration and the static pressure stations.

3.2 Remote Computer Setup

A remote computer network was established under this project. An IBM personal computer system 2, model 50 with a modem (Fastalk) was choosen as remote terminal to connect to the EADS (Engineering And Design System) of super computer CRAY-X/MP AT NASA/MSFC. The terminal emulator VTEK allows IBM PS/2 to emulate DEC VT100 terminal as well as Tektronic 4105/4014/4010 terminals. Figure 2 shows the arrangement through the communication network PSCN.

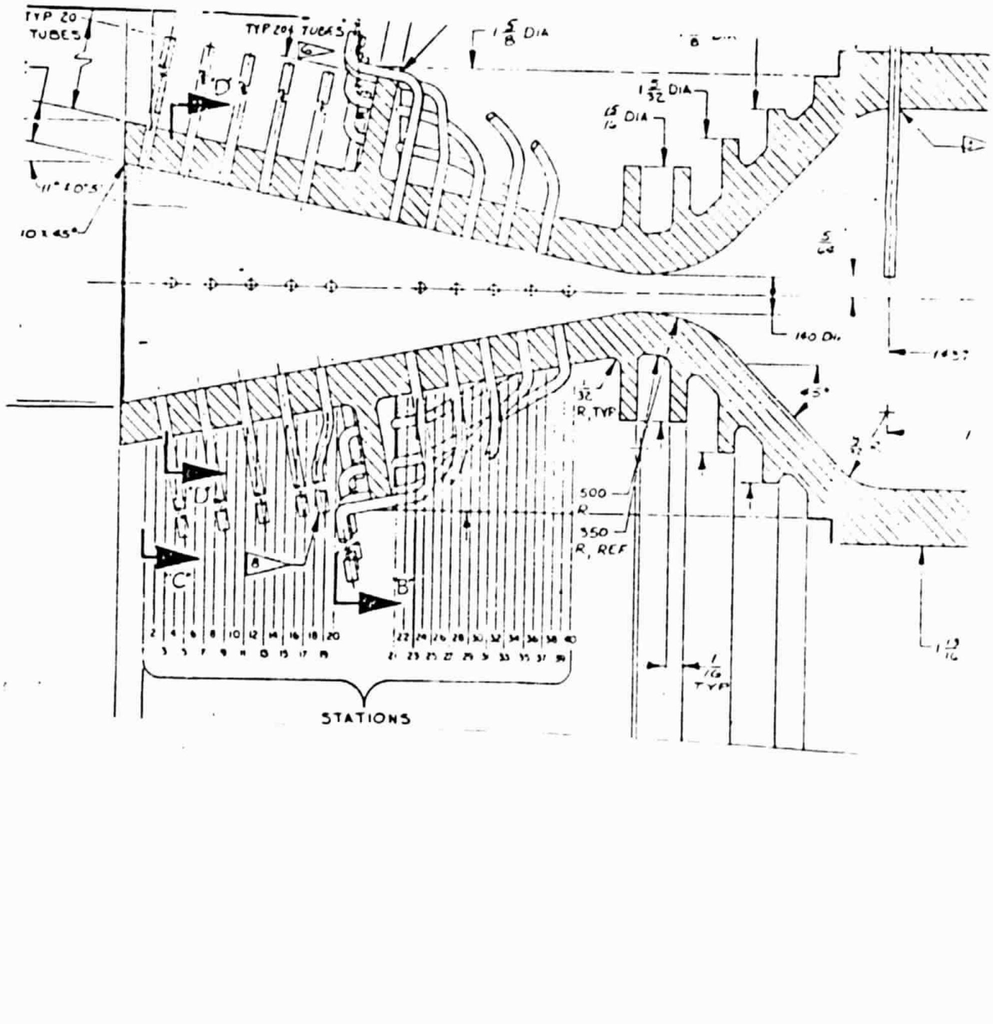

Figure 1. NOZZLE CONFIGURATION

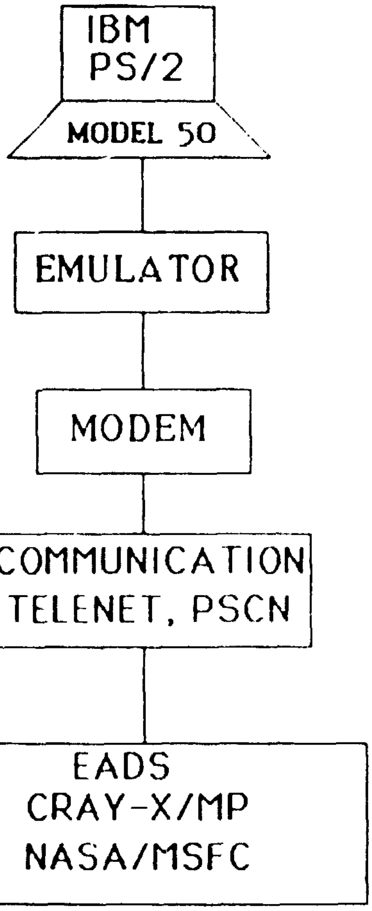

Figure 2. REMOTE COMPUTER WORK STATION SETUP

3.3 Grid Generation

The first step to calculating the flow in a nozzle is to generate an appropriate computational grid. Two grid generator codes have been used. The first grid generator is called TBGG and was developed by Smith and Wiese [2] at Langley Research Center. This grid generator was run on the VAX System in NASA/MSFC. It is interactive in the Tektronic 4010 mode and can be used to generate body fitted grids algebracally. The output file then was copieo to a magnetic tape and read onto the EADS System. The second grid generator is called GENIE and was developed by Soni [3] . GENIE code can provide computational grids for wide range of geometries related to internal flow problems. The process uses several techniques either separately or in combination to generate grids quickly and economically for arbitrary geometries. GENIE code was adopted later in this project due to it's more generic structure.

Computations were performed on the nozzle configuration, the major geometric parameters for the nozzle and upstream plenum chamber are shown in Figure 3. This geometry was introduced into the grid generator as a discrete number of dimensional parts or as a set of equations. The exponential packing function was used to cluster the grids near the walls as shown in Figure 4. The blow up of grids near the nozzle throat region is shown in Figure 5.

3.4 Governing Equations

Two-dimensional Navier-Stokes equations in fully conservative form are used to describe the nozzle flow. The transformed equations can be written as

$$U_\tau + E_\xi + F_\eta + H = \frac{1}{R_e} (R_\xi + S_\eta + T)$$

Where

$$U = \bar{U}/J$$
$$E = (\xi_t \bar{U} + \xi_x \bar{E} + \xi_y \bar{F})/J$$
$$F = (\eta_t \bar{U} + \eta_x \bar{E} + \eta_y \bar{F})/J$$
$$H = (\bar{F} + \bar{H})/(yJ)$$
$$R = (\xi_x \bar{R} + \xi_y \bar{S})/J$$
$$S = (\eta_x \bar{R} + \eta_y \bar{S})/J$$
$$T = (\bar{S} + \bar{T})/(yJ)$$

And the curvilinear coordinate system ξ, η and τ are represented as

$$\tau = t, \quad \xi = \xi\ (t,x,y), \quad \eta = \eta\ (t,x,y)$$
$$\xi_t = (x_\eta\, y_\tau - y_\eta\, x_\tau)J\ , \quad \eta_t = (y_\xi\, x_\tau - x_\xi y_\tau)J$$
$$\xi_x = y_\eta J \quad , \quad \eta_x = -y_\xi J$$
$$\xi_y = -x_\eta J \quad , \quad \eta_y = x_\xi J$$
$$J^{-1} = x_\xi\, y_\eta - y_\xi\, x_\eta\ .$$

The vectors in the above equations are

$$\bar{U} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ e \end{bmatrix}, \quad \bar{E} = \begin{bmatrix} \rho u \\ p+\rho u^2 \\ \rho uv \\ (e+p)u \end{bmatrix}, \quad \bar{F} = \begin{bmatrix} \rho v \\ \rho uv \\ p+\rho v^2 \\ (e+p)v \end{bmatrix}, \quad \bar{H} = \begin{bmatrix} 0 \\ 0 \\ -p \\ 0 \end{bmatrix}$$

$$\bar{R} = \begin{bmatrix} 0 \\ \tau_{xy} \\ \tau_{xy} \\ \frac{\gamma}{Pr}\kappa c_x + u\ \tau_{xx} + v\ \tau_{xy} \end{bmatrix}, \quad \bar{S} = \begin{bmatrix} 0 \\ \tau_{xy} \\ \tau_{yy} \\ \frac{\gamma}{Pr}\kappa\varepsilon_y + u\ \tau_{xy} + v\ \tau_{yy} \end{bmatrix}, \quad \bar{T} = \begin{bmatrix} 0 \\ 0 \\ -\tau_{\phi\phi} \\ 0 \end{bmatrix}$$

The quantities $\tau_{xx}, \tau_{xy}, \tau_{yy}$ are components of the stress tensors, γ is the ratio of specific heat, k is the coefficient of thermal conductivity, Pr is the Prandtl Number and Re is the Reynolds Number. In order to complete the set of governing equations, the equation of state

$$P = \rho RT$$

is used, where ρ is the density, R is the gas constant and T is the absolute temperature.

3.5 Navier-Stokes Solver

The Navier-Stokes code, PARC2D, is a modification of the ARC2D code that was developed by Pulliam and Steger [4] at NASA/Ames Research Center. It uses a thin layer approximation to the Navier-Stokes equations with parabolized viscous stress terms. It is written in the strong conservative form in curvilinear coordinates. The non-iterative implicit approximate factorization scheme of Beam and Warming [5],[6] is used with a fourth order artifical dissipation.

The PARC2D code is a modification of the ARC2D code by Cooper [7] at Sverdrup Technology, Inc./AEDC Group which removed the thin layer approximation. It is fully elliptic, requiring closed boundary conditions and an initial condition everywhere in the flow field. The code is modular and fully vectorized. It assumes that the gas is perfect gas with constant gamma and constant Prandtl number, and the Sutherland viscosity law for the temperature variation of viscosity.

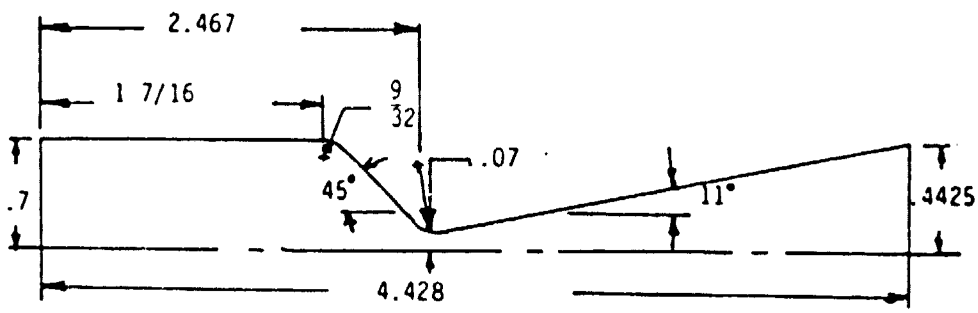

Figure 3. GEOMETRY OF CONICAL NOZZLE AND PLENUM CHAMBER
(ALL DIMENSIONS ARE IN INCHES)

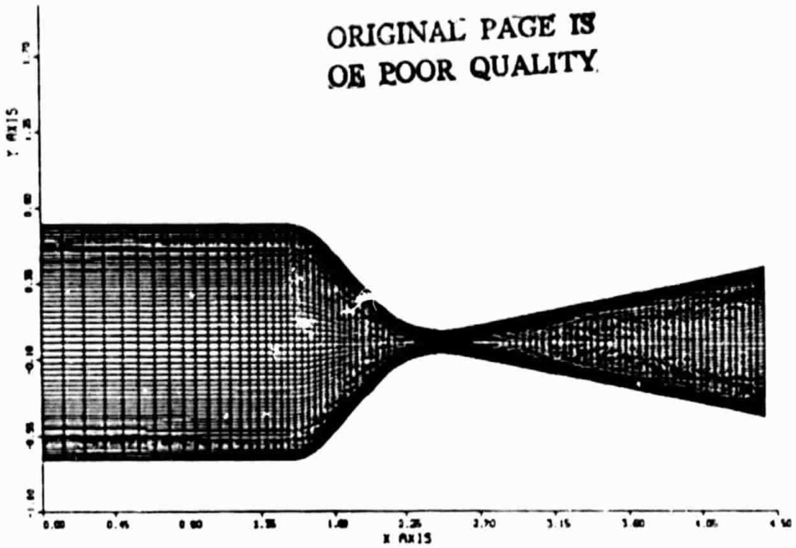

FIGURE 4. 90X81 GRID APPLIED TO THE CONICAL NOZZLE

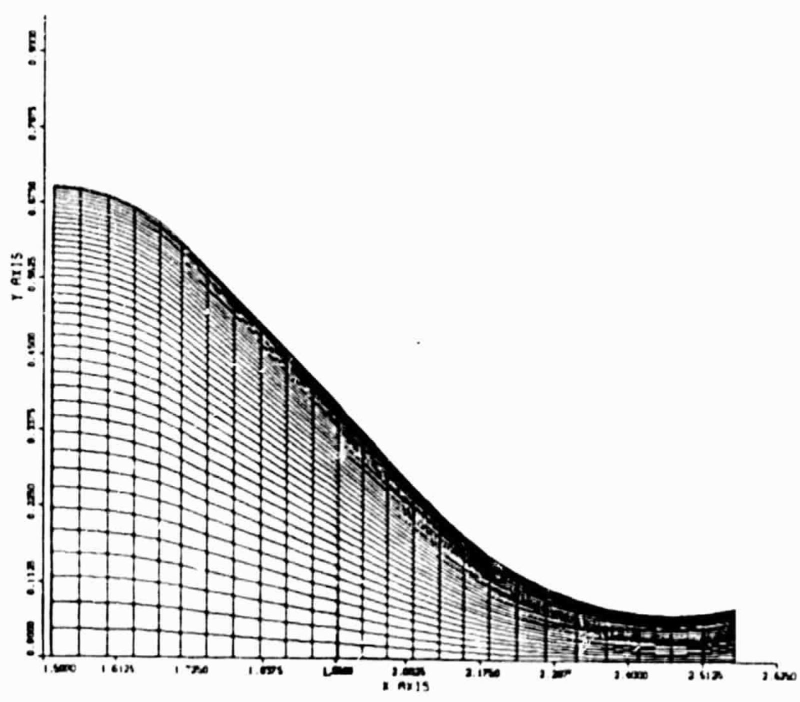

FIGURE 5. BLOW-UP OF GRID 90X81 GRID NEAR NOZZLE THROAT REGION

3.6 Initial and Boundary Conditions

For the present calculations the stagnation conditions were taken as reference and the reference length was one inch, since the grid coordinates were calculated in inches. The Reynolds number is based upon a reference sound speed and length. As suggested by Collins [8] that the fastest convergence occurred by taking $u = v = o$ and density and energy equal to their stagnation values. These values were assigned initially throughout the flow field.

Other options used are as follows: adiabatic wall, axisymmetric, viscous, laminar. Free boundaries (allowing inflow or outflow) were assumed at the upstream and downstream boundaries, and the other boundaries were axis of symmetry and no slip/adiabatic wall.

Boundary conditions must be imposed upon all of the boundaries. For the free boundaries this consists of a specification of the pressure. The stagnation pressure was imposed upstream but the downstream boundary condition posed a problem. For flows with strong viscous/inviscid interaction the pressure cannot be expected to be constant across nozzle and greater pressure variations would exist across the exit of a conical nozzle. However, a constant downstream pressure must be specified. It is usually prudent to specify a pressure somewhat lower than the minimum expected downstream pressure. No boundary conditions are required on the axis nor on the wall. If constant temperature wall conditions are assumed then the temperature must be specified. Note that the entire wall does not have to be assumed to be at the same temperature.

An efficient convergence procedure for these nozzle problems was also suggested by Collins. The steps are as follows:

1) Initially set $q = 0$, $\bar{\rho} = 1$ and $\bar{E} = 1/\gamma(\gamma - 1)$ everywhere in flow field.

2) Set initial parameters as follows: DIS2 = 0.2, DIS4 = 0.35, PCQMAX = 10.0, DTCAP = 5.0. Run until the axial velocity is positive everywhere in the plenum chamber.

3) Slowly reduce DIS4 to 0.25 (all other parameters constant).

4) Slowly reducde DIS2 to 0.00.

5) Check DT and set DTCAP to about one-half of the minimum DT for the last series of interations. Then run until L2 reaches an acceptalbe value (10^{-8} to 10^{-9}).

In this discussion DIS4 and DIS2 are parameters related to the fourth order and second order dissipation, respectively, PCQMAX sets the maximum change in any variable during an iteration, DT is the time step and DTCAP is the maximum allowable time step. L2 is a convergence measure.

4. RESULTS

All the computations were made on the CRAY-X/MP super computer at NASA/MSFC. A grid of 90x81 was used in the calculations which required approximately 800k computer storage. Convergence occurred after 7000 iterations. Computations were made using the test conditions that corresponded to the MSFC in-house measurements on the nozzle described in Figure 1 The test gas was CO_2, stagnation temperature was 1060°R and the stagnation pressure had three values, 29.4, 14.7 and 7.4 psia. Since only wall pressure measurements were made, that was the only parameter that could be used for comparison with the computations. The computational code assumed a constant gamma whereas the gamma for CO_2 varied from about 1.22 in the plenum chamber to 1.38 at the exit as shown in Figure 6. Therefore, exact comparison could never be expected to occur between the measurements and computations.

The calculated results was illustrated by the PLOT3D, which is a commercial software package to plot the flow field distributions. PLOT3D is available at NASA/MSFC which can be run on either EADS or IRIS workstation. The flow parameters, such as pressure, temperature, velocity and Mach number can be plotted as contours on the nozzle crossection. In order to illustrate some particular parameters at a specific location (such as the Mach number along the center line), some specific plotting programs were developed by using graphical software package DISSPLA to fit these needs.

Comparisons of computed and measured wall pressure at different chamber pressures are given in Figures 7, 8 and 9 respectively. The agreements at each stagnation pressure are quite well. It is difficult to distinguish the calculated and measured results from these figures. However, it shows some differences near the nozzle exit in the semi-log scale in Figure 10, which is drawn to magnify the differences. Several reasons are possible for the lack of agreement, in addition to the need for a variable gamma computational code. These include the possibility that the adiabatic wall boundary condition is not applicable and computations at a constant wall temperature should be performed to examine that possibility. In addition, there is a strong possibility that condensation of the CO_2 was occurring during the expansion and agreement would not be expected until the stagnation temperature was raised to eliminate all possibility for condensation. Finally, at the exit the Knudsen number is about 0.06, based on a mean free path at wall conditions and a crude estimate of the displacement thickness. Slip is expected to occur under such circumstances and the slip wall boundary condition formulated by Collins [9] will be implimented.

Typical profiles of Mach number and static temperature across the nozzle at a location near the nozzle exit are shown in Figures 11 and 12. The large temperature gradients near the wall are caused by the assumed adiabatic wall condition.

Figure 13 shows the Mach number along the center line. The Mach number increases rapidly after passed the throat region. The contour plots of Mach number, Pitot Pressure, Temperature and Velocity in Figures 14, 15, 16 and 17 are presented to show the computational features of the entire field. These contour plots clearly show the phenomena of strong viscous/inviscid interaction along the nozzle wall, especially near the nozzle exit. Figure 18 shows the velocity-vector field on the upper half of the nozzle. It is seen that the boundary layer thickness is increasing along the nozzle flow stream. These results show that the present code predicts the low thrust nozzle quite well except near the nozzle exit.

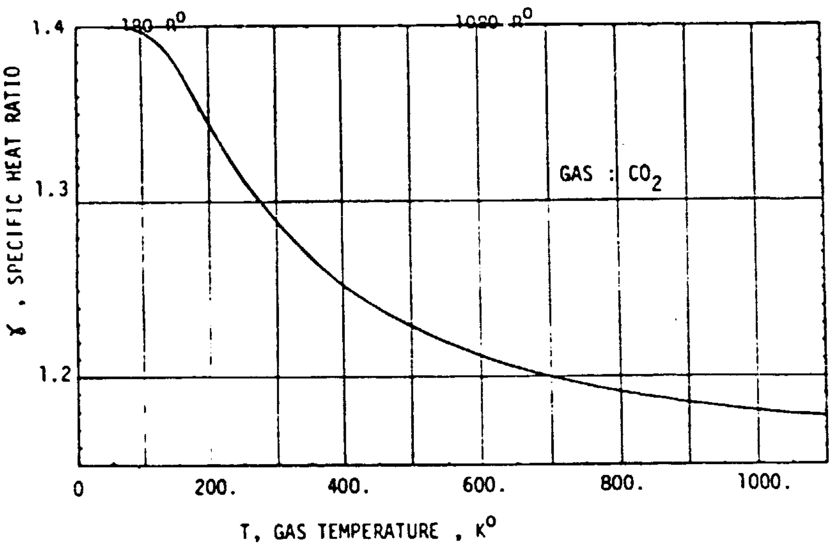

Figure 6. THE SPECIFIC HEAT RATIO OF CO_2 GAS

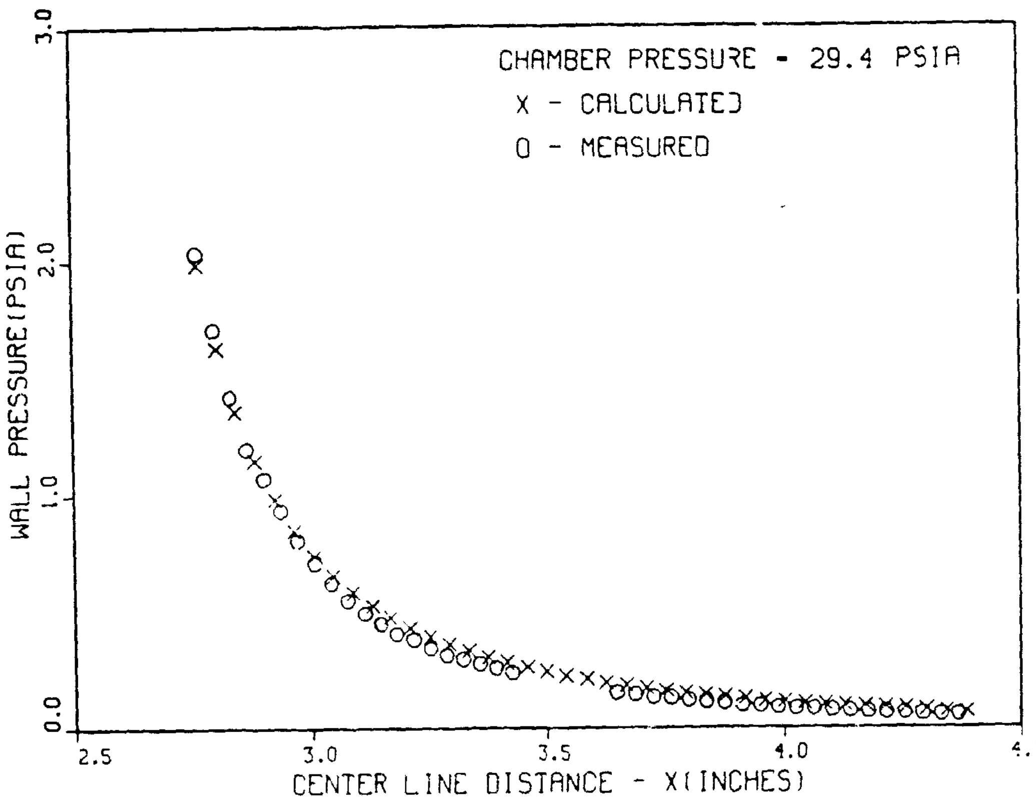

Figure 7. WALL PRESSURE VS CENTER DISTANCE

(CHAMBER TEMPERATURE=1060°R,

CHAMBER PRESSURE=29.4 psia)

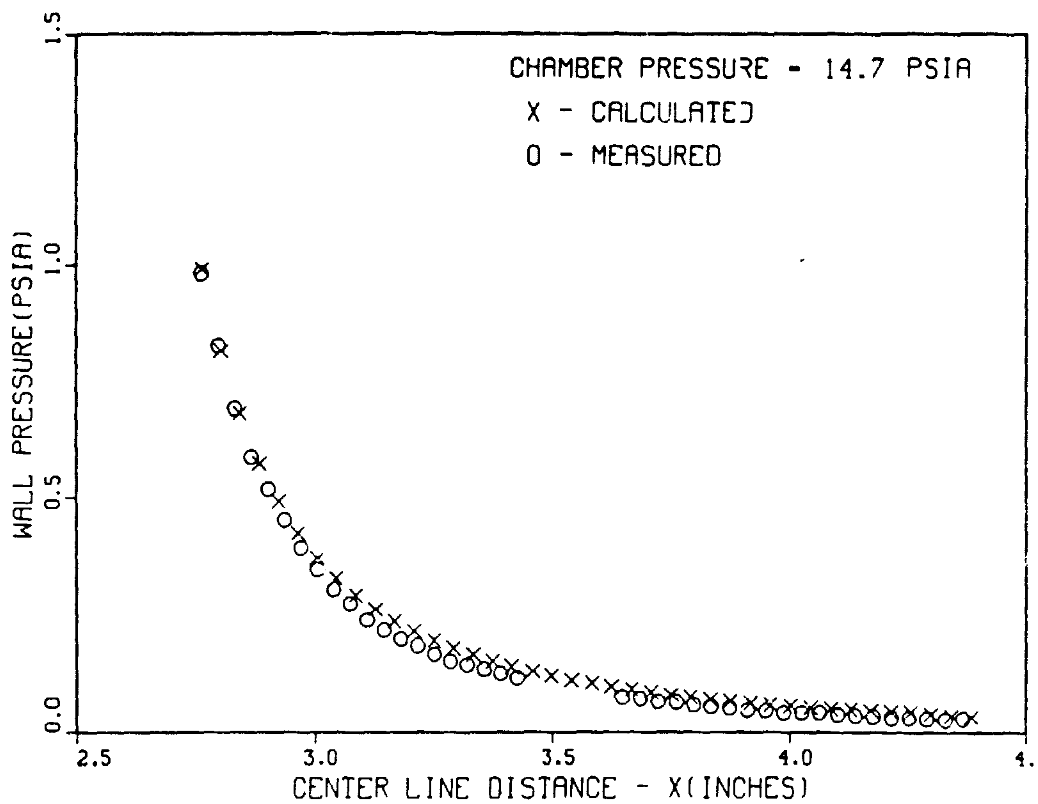

Figure 8. WALL PRESSURE VS CENTER DISTANCE
(CHAMBER TEMPERATURE=1060^{0}R,
CHAMBER PRESSURE=14.7 psia)

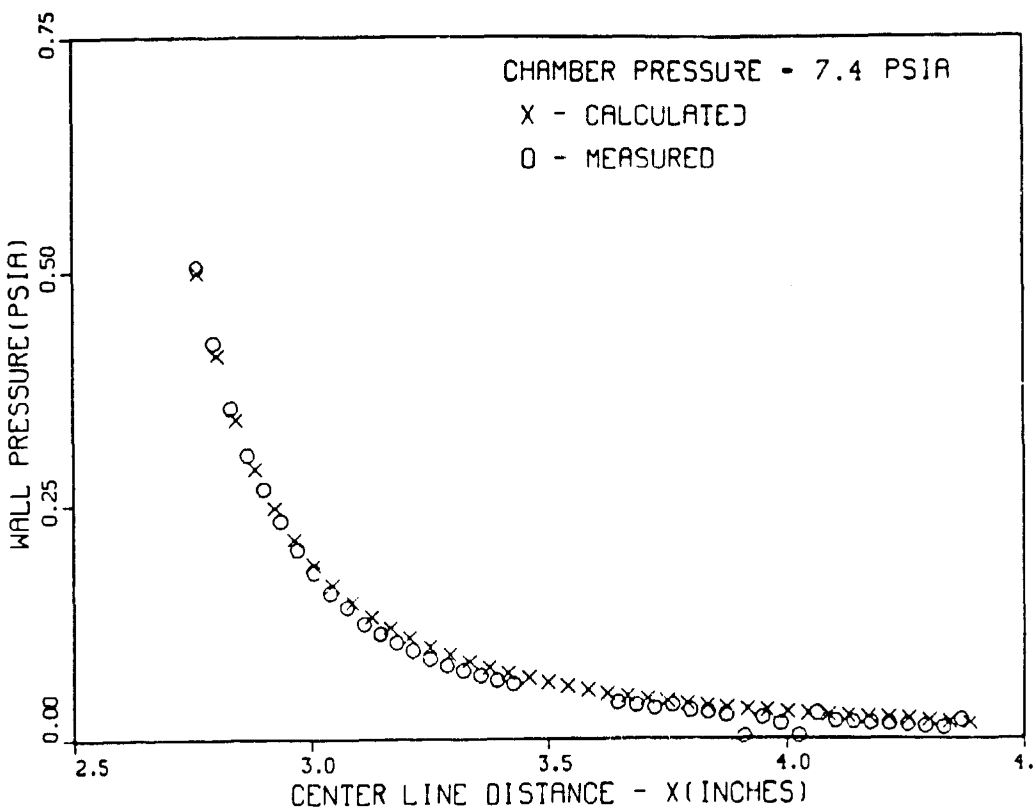

Figure 9. WALL PRESSURE VS CENTER DISTANCE
(CHAMBER TEMPERATURE=1060^{0}R,
CHAMBER PRESSURE=7.4 psia)

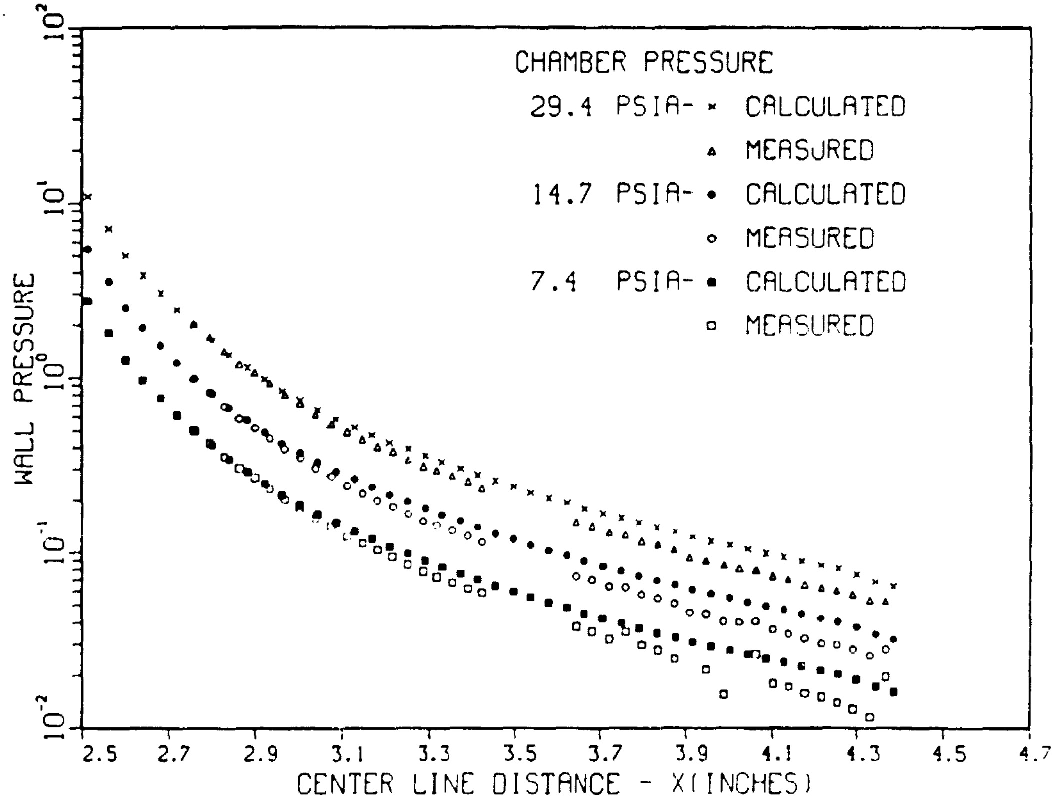

Figure 10. WALL PRESSURE VS CENTER DISTANCE

(CHAMBER TEMPERATURE = 1060^{0}R)

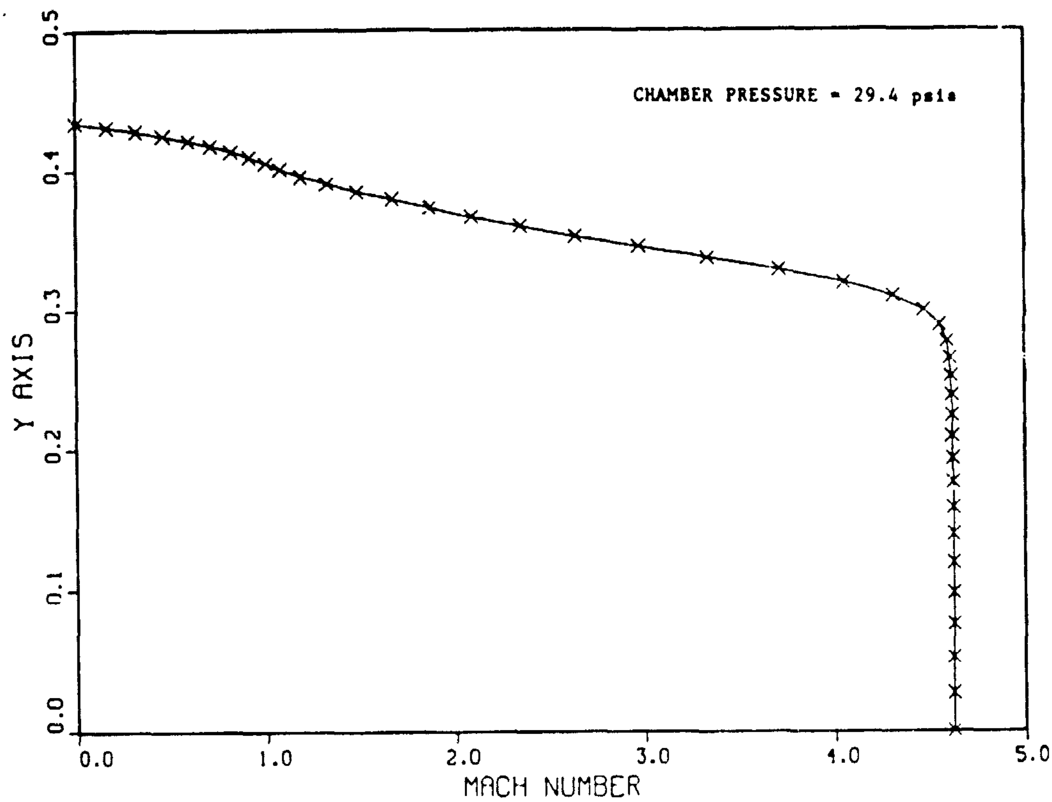

Figure 11. MACH NUMBER VS RADIAL DISTANCE
(CHAMBER TEMPERATURE=1060°R,
CHAMBER PRESSURE= 29.4 psia)

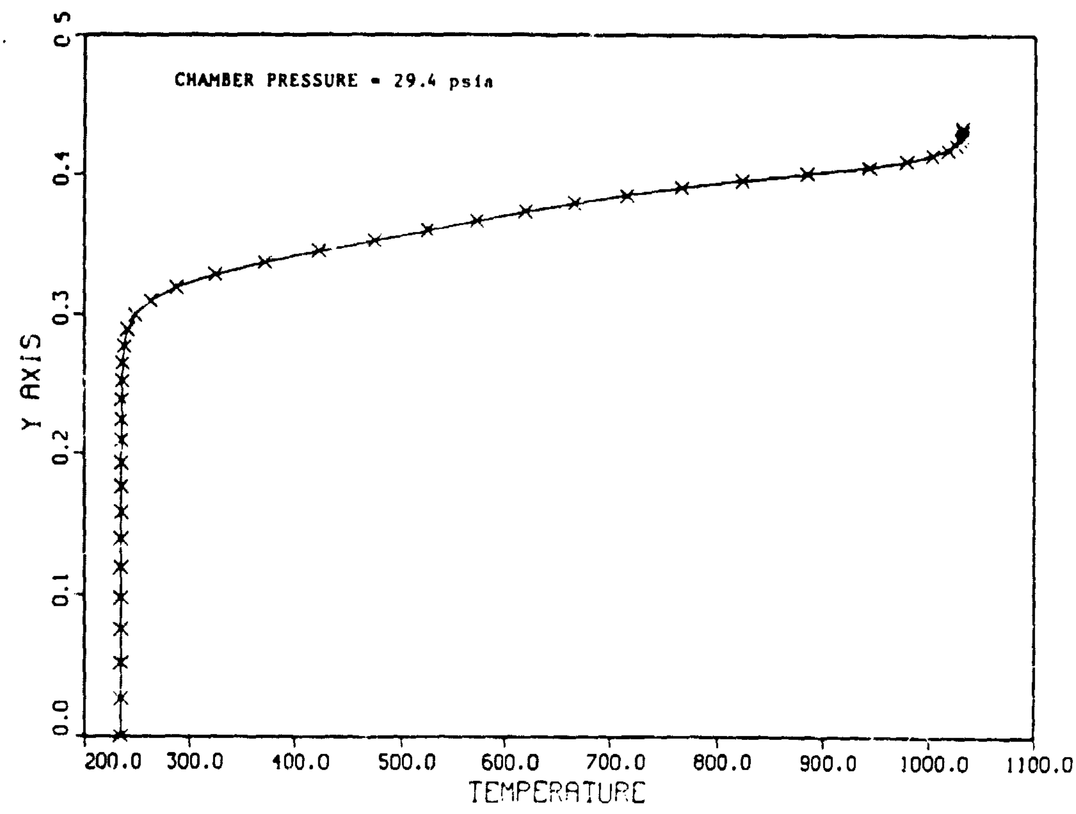

Figure 12. TEMPERATURE VS RADIAL DISTANCE

(CHAMBER TEMPERATURE = 1060°R,

CHAMBER PRESSURE = 29.4 psia)

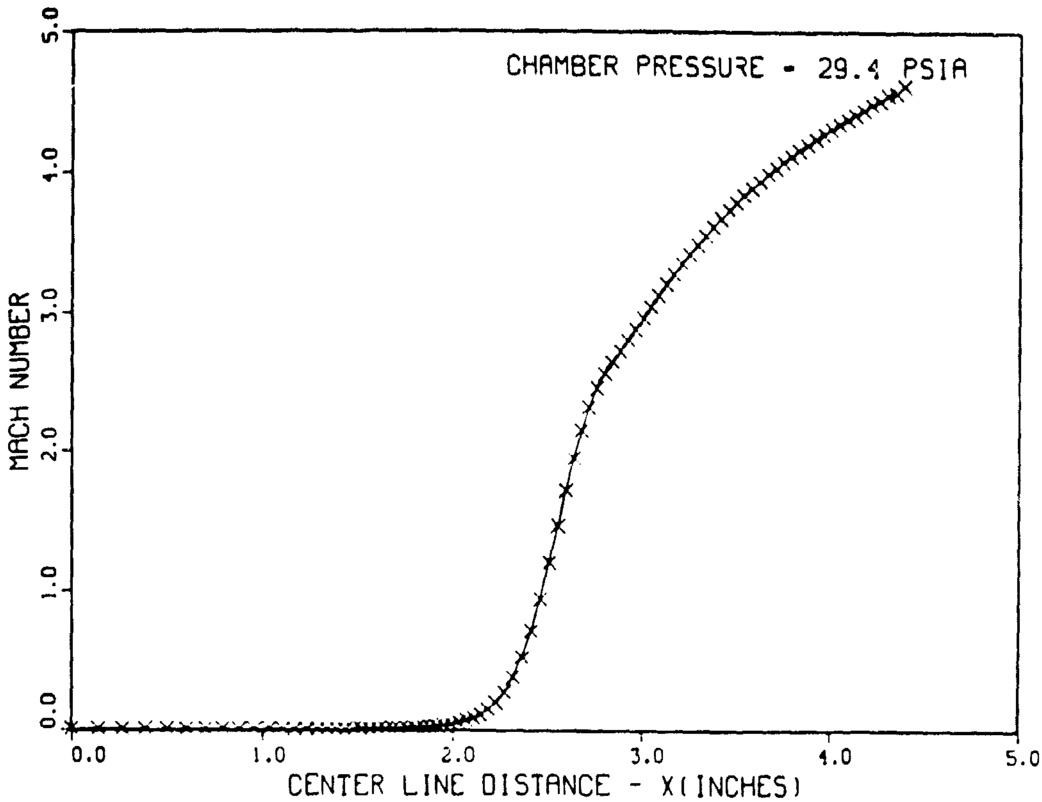

Figure 13. MACH NUMBER VS CENTER DISTANCE
(CHAMBER TEMPERATURE=1060^{0}R,
CHAMBER PRESSURE= 29.4 psia)

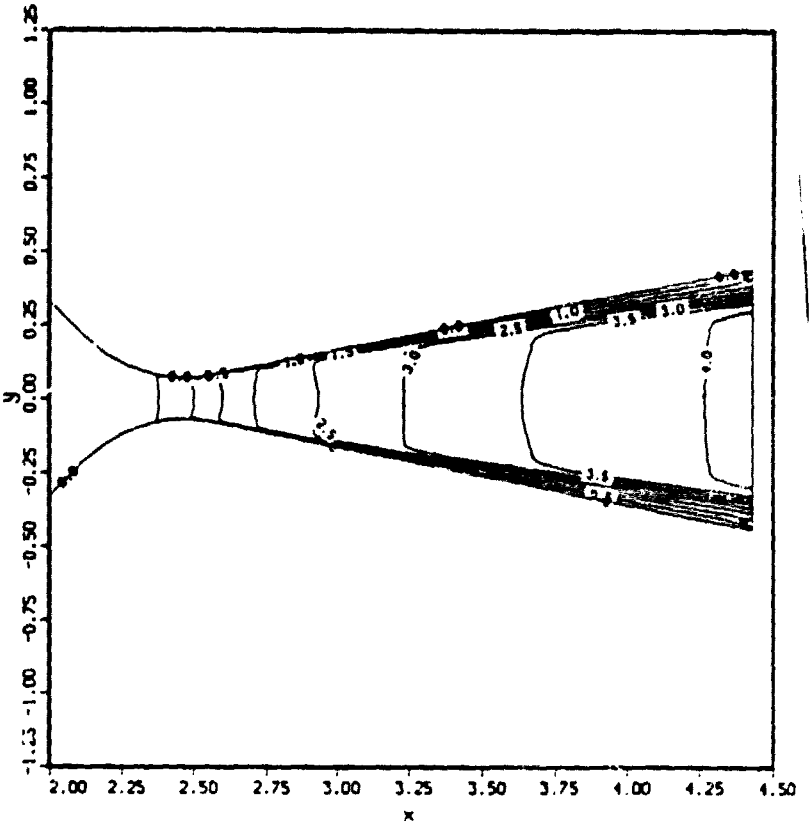

Figure 14. MACH NUMBER CONTOUR LINES

(CHAMBER TEMPERATURE=1060^{0}R,

CHAMBER PRESSURE= 29.4 psia)

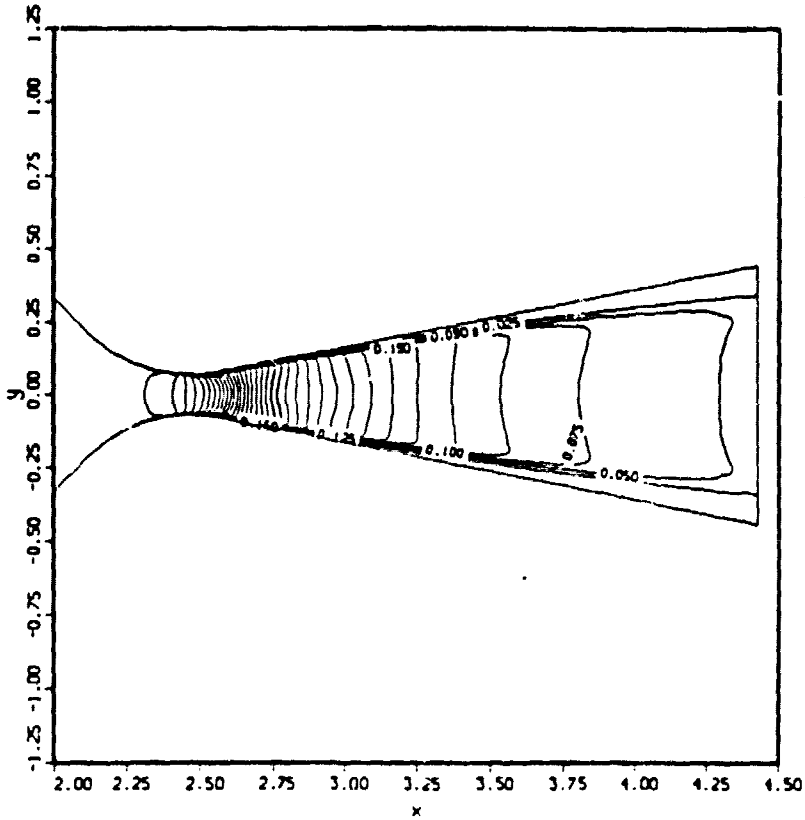

Figure 15. PITOT PRESSURE CONTOUR LINES
(CHAMBER TEMPERATURE=1060^{0}R,
CHAMBER PRESSURE= 29.4 psia)

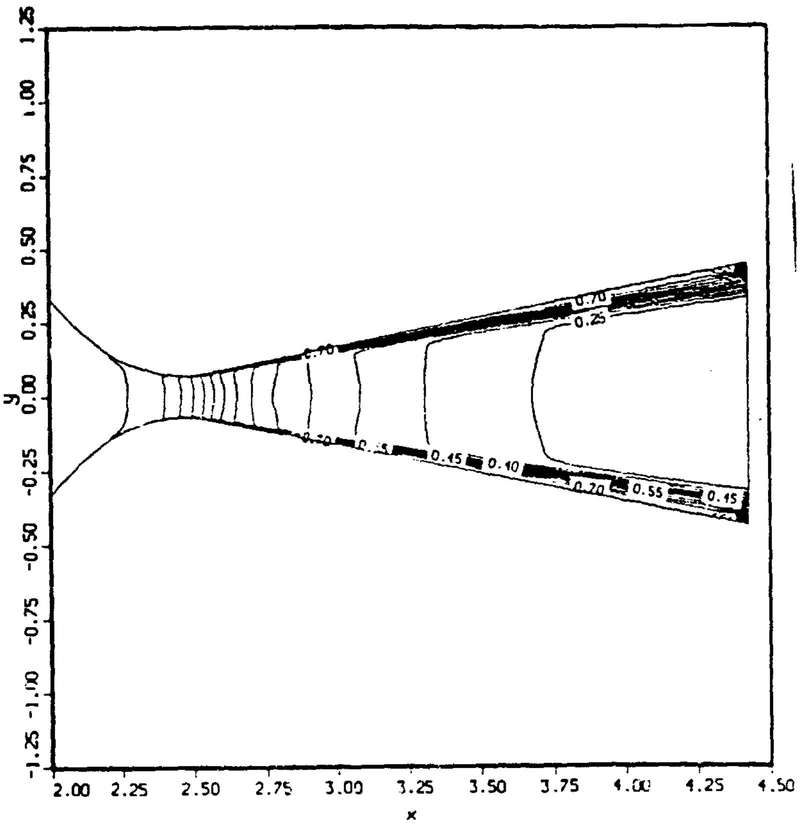

Figure 16. TEMPERATURE CONTOUR LINES
(CHAMBER TEMPERATURE=1060°R,
CHAMBER PRESSURE= 29.4 psia)

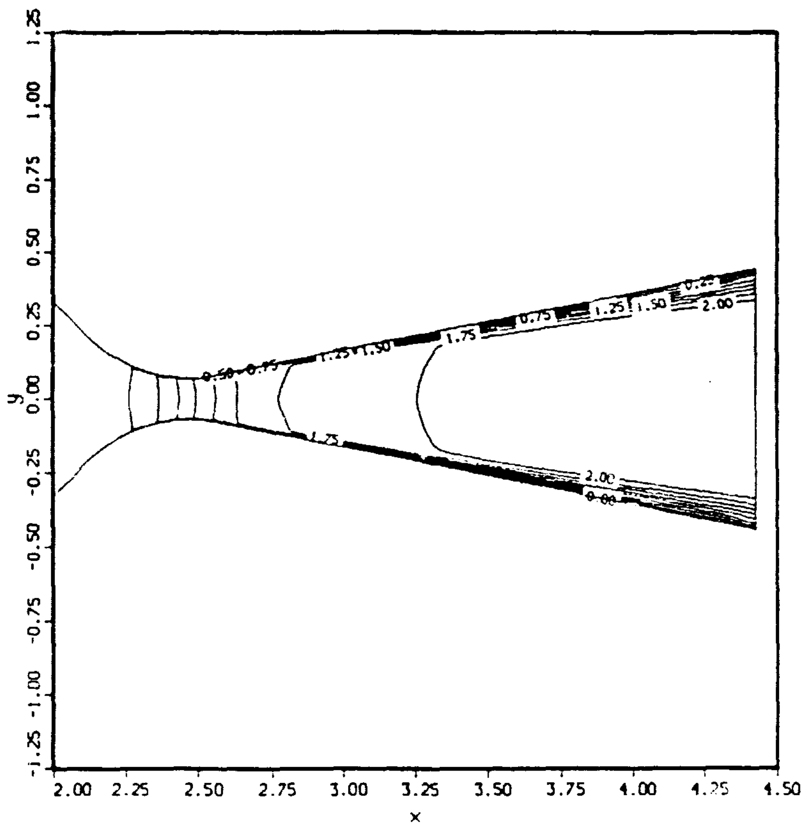

Figure 17. HORIZONTAL VELOCITY CONTOUR LINES

(CHAMBER TEMPERATURE=1060°R,

CHAMBER PRESSURE= 29.4 psia)

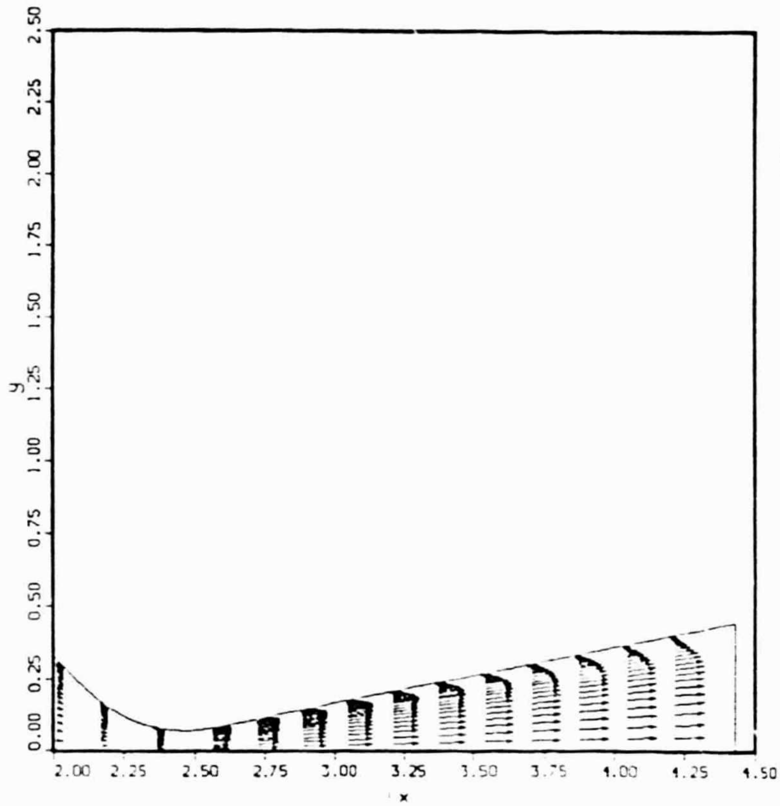

Figure 16 VELOCITY VECTOR
(CHAMBER TEMPERATURE = 1060^0R,
CHAMBER PRESSURE = 29.4 psia)

5. CONCLUSIONS AND FUTURE WORKS

The following conclusions have been made from the results of this study.

1) The grid generator code GENIE provided a good means generating useful computational grids for the Navier-Stokes computations of the nozzle flow.

2) The PARC2D code yields reasonable solutions for the flow field in supersonic nozzles at low Reynolds numbers where large viscous/inviscid interaction exists. Because the code is modular it can be easily modified.

A number of works will be done in the second year :

1) The variable gamma to the PARC2D code will be implemented.

2) The code will include the slip boundary conditions.

3) In addition to the conical nozzle, the calculation will be performed on the sonic nozzle flow field and the plume, which is currently under testing in AEDC. The configuration is shown in Figure 19 and the computational grid has been developed as shown in Figure 20. This will be one of the tasks in the next year.

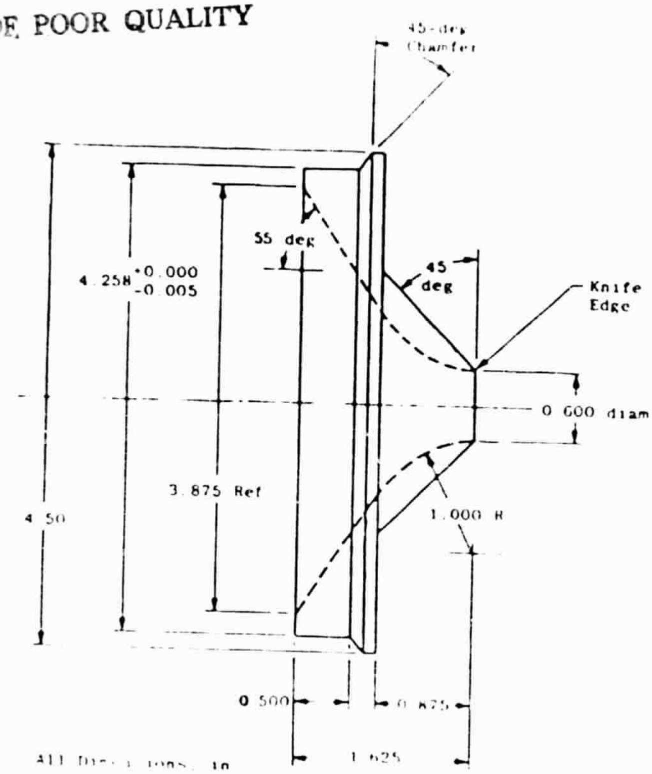

Figure 19. CONFIGURATION OF SONIC NOZZLE

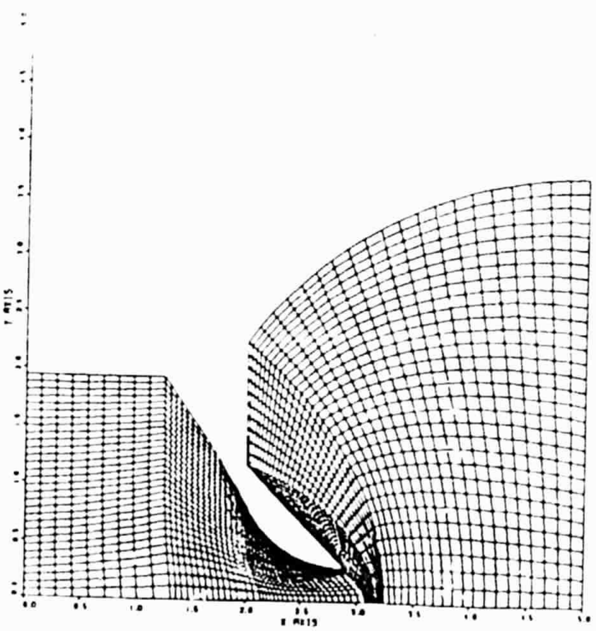

Figure 20. 90x61 GRID APPLIED TO THE SONIC NOZZLE

6. ACKNOWLEDGMENTS

This work is supported by NASA/Marshall Space Flight Center under contract NAG8-064 through the Historical Black College and University program. I would like to thank Dr. Lynn C. Chou and his colleagues of the Computational Fluid Dynamics Branch/Aerophysics Division/System Dynamic Laboratory for their assistances in this research work, especially the nozzle test results and computer utilizations.

I also like to express my appreciation to Mr. Eric Tzeng, the student research assistant, for his help in the computer setup and computer drawings.

7. REFERENCES

1. Chou, L. C., and Carter, J. A., "Analytical and Experimental Investigation of a CO_2 Gas Stream Discharging Through a Conical Nozzle Into a Vacuum Tank," MSFC MEMO, July 31, 1984.

2. Smith, R. E. and Wiese, M. R., "Interactive Algebraic Grid-Generation Technique." NASA Technical Paper 2533, 1986.

3. Soni, B. K. "Two and Three Dimensional Grid Generation for Internal Flow Applications of Computational Fluid Dynamics." AIAA-1526-85, 1985.

4. Pulliam, T. H., "Euler and Thin lager Navier-Stokes Codes: ARC2D, ARC3D," Notes for computational Dynananics Users Workshop, UTSI E02-4005-023-84, March, 1984

5. Warming, R. F. and Beam, R. M, "On the construction and Application of Implicit Factored Schemes for Conservation Laws," SIAM-AMS Proceedings, Vol II, pp 85-129, 1970

6. Beam, R. M. and Warming, R. F., "An Implicit Finite-Difference Algorithm for Hyperbolic Systems in Conservation Law Form," Journal of Computational Physics, Vol 22, pp 87-110, September 1976.

7. Cooper, G. K. Sverdrup Technology, Inc/AEDC, "The PARC Codes-Users Manual." September, 1986.

8. Collins, F. G., "Computation of Nozzle Flow Fields using the PARC2D Navier-Stokes Code," unpublished report. NASA/MSFC, August 1986.

9. Collins, F. G., "Low Density Gas Dynamic Wall Boundary Conditions," unpublished report, NASA/MSFC, September, 1985.