

To the Graduate Council:

I am submitting herewith a thesis written by Glenn Douglas Garrard entitled "Calibration of an Axisymmetrical Navier Stokes Computational Fluid Dynamics Program for Predicting Steady State Diffuser Performance." 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 in Mechanical Engineering.

Roy J. Schulz
Roy J. Schulz, Major Professor

We have read this thesis and recommend its acceptance:

Roger Crawford
R. L. Young

Accepted for the Council:

Lawminkel
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 Master's degree at The University of Tennessee, Knoxville, I agree that the Library shall make it available to borrowers under the rules of the Library. Brief quotations from this thesis are allowable without special permission, provided that accurate acknowledgment of the source is made. Permission for extensive quotation from or reproduction of this thesis may be granted by my major professor, or in his absence, by the Head of Interlibrary Services when, in the opinion of either, the proposed use of the material is for scholarly purposes. Any copying or use of the material in this thesis for financial gain shall not be allowed without my written permission.

Signature

Glenn Douglas Garrard

Date

April 18, 1990

CALIBRATION OF AN AXISYMMETRICAL NAVIER STOKES
COMPUTATIONAL FLUID DYNAMICS PROGRAM
FOR PREDICTING STEADY STATE DIFFUSER
PERFORMANCE

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

Glenn Douglas Garrard

May 1990

DEDICATION

This thesis is dedicated to my wife, Angelia Gail Garrard, whose support and encouragment has enabled me to obtain this goal in my life.

ACKNOWLEDGMENTS

I would like to acknowledge the United States Air Force and Sverdrup Technology, Inc. for support provided at the Arnold Engineering Development Center. I would also like to express my appreciation to the many individuals who have assisted and supported me throughout my studies. Dr. G. K. Cooper and Mr. W. J. Phares are due a special thanks as they are the ones who introduced me to the field of computational fluid dynamics. Appreciation is also extended to Dr. R. J. Schulz for his guidance and technical support and to the entire review committee for their constructive review comments. Mr. L. B. Bates deserves special recognition for his assistance with the diffuser material.

The work reported herein was conducted at the Arnold Engineering Development Center (AEDC), Air Force Systems Command (AFSC). The results were obtained by Sverdrup Technology, Inc., AEDC Group, operating contractor for the propulsion test facilities at AEDC, AFSC, Arnold Air Force Base, Tennessee, under Project DB84EW. The Air Force Project Manager was Captain Mark Briski. The manuscript was submitted for publication on April 27, 1990.

ABSTRACT

The calibration of the axisymmetrical Navier-Stokes computational fluid dynamics (CFD) program PARC2D for predicting steady state diffuser performance is presented. The calibration effort used data from available literature and was limited to an axisymmetrical diffuser system that used single specie, single stream, nonreactive flow. The calibration range included a convergent/divergent nozzle with an exit area to throat area ratio of 9, a diffuser with a diffuser area to nozzle throat area ratio of 36 with a diffuser length to diameter ratio of 4, and a diffuser with a diffuser area to nozzle throat area ratio of 16 with a diffuser length to diameter ratio of 12. Calibration parameters selected for the study were diffuser pressure rise ratio and diffuser wall static pressure. The calibration effort was divided into three parts. First, a parametric study was conducted to observe the relative effects of various perturbations in the solution technique on the calculated flow field. Variations in the cell volume, the grid resolution in the diffuser, the initial conditions used to start the solution, and the turbulence mixing level were studied. Secondly, the PARC2D program results were compared to data for the selected diffuser configurations. Finally, a modification

to the program which based the local intensity of turbulence in the flow on the local Mach number was made and the results were compared to data.

The parametric study results demonstrated the effects of variations in the solution technique, and provided parameter sensitivity information that was used in the remainder of the research. The cell volume study showed that the computational cell volume could be reduced to half of the actual test hardware volume with no adverse effects. Reducing the cell volume to a minimum, however, caused a lower cell pressure to be obtained. Increasing the resolution of the grid in the diffuser by including 7,000 additional grid points did not provide any significant benefits, but increased the computer memory requirements by 50 percent. The initial condition study concluded that using a cell pressure in the initial conditions lower than the expected result provided a faster convergence rate than a higher cell pressure, and that filling the diffuser with supersonic flow provided approximately the same convergence rate as specifying that there was not any flow at all in the diffuser. The turbulence mixing level study showed the effect of varying the level of the turbulence viscosity on the solution. As the level of turbulent viscosity was increased, the final value of the calculated cell pressure was lowered.

In the actual calibration of the PARC2D program to test data, the PARC2D program predicted diffuser pressure rise ratio to within 5 percent of the test data for started diffuser operation. The wall static pressures, however, indicated that the PARC2D program overpredicted the shock strength in the diffuser. For conditions where the diffuser was unstarted and the nozzle flow was separated from the nozzle wall, the PARC2D cell pressure predictions were approximately 30 to 60 percent higher than the test data. The qualitative trend of the diffuser wall static pressures was correct, but quantitatively the pressures were too high because of the incorrect cell pressure. For unstarted diffuser operation with the nozzle flowing full, a stable solution was not obtained. The calculated cell pressure varied in a cyclic nature about the test data with a deviation of ± 30 percent. Because of the large number of iterations required to complete the convergence, final resolution of the predicted cell pressure was not possible.

The modification to the algebraic turbulence model was promising in that the largest time step size that permitted a stable solution increased by a factor of five for the cases considered. The modification did not, however, provide the correct diffuser pressure rise ratio. For the two cases operating with the diffuser in the started mode,

the modified algebraic turbulence model resulted in a cell pressure as much as 80 percent higher than the test data. The diffuser wall static pressures showed that the shock strength in the diffuser was overpredicted. The one unstarted diffuser case considered did not reach a stable solution and the cell pressure was as much as 60 percent higher than the test data.

TABLE OF CONTENTS

SECTION	Page
1. INTRODUCTION	1
1.1 BACKGROUND	4
1.2 CALIBRATION DATA	7
1.3 COMPUTATIONAL FLUID DYNAMICS PROGRAM	10
1.4 STATEMENT OF PROBLEM	12
2. LITERATURE SURVEY	14
3. CALIBRATION OF THE PARC2D PROGRAM	22
3.1 GENERAL	22
3.2 PARAMETRIC STUDY	29
3.3 AERODYNAMIC CALIBRATION	40
3.4 MODIFICATION STUDY	48
4. SUMMARY AND CONCLUSIONS	54
5. RECOMMENDATIONS	63
BIBLIOGRAPHY	66
APPENDIXES	71
A. FIGURES	72
B. PARC2D THEORY	147
VITA	152

LIST OF FIGURES

FIGURE	Page
1. Typical turbine engine test installation	73
2. Typical rocket motor test installation	73
3. Parameters of interest for diffuser systems	74
4. Graphical presentation of diffuser performance	75
5. Flow in the diffuser for different operating conditions	76
6. Effect of secondary flow on diffuser performance	77
7. Test apparatus used to obtain calibration data	78
8. Experimental performance of cylindrical diffuser for various diffuser to nozzle throat area ratios (A_d/A_t) and diffuser length to diameter ratios (L/D)	79
9. Wall static pressures for 1.5 inch diameter, 6 inch length diffuser at various total pressures (P_t)	80
10. Wall static pressures for 1.0 inch diameter, 12 inch length diffuser at various total pressures (P_t)	81
11. Subscale experimental configuration	82
12. Tube surface static pressure (P_{ts}) versus distance from nozzle exit plane (x) for subscale test	83
13. Cell wall static pressure (P_{cw}) versus distance from nozzle exit plane (x) for subscale test	84

FIGURE	Page
14. Fan cowl surface pressure (P_{fan}) versus distance from fan stream exit plane (x) for engine/test cell	85
15. Core plug surface pressure (P_{cp}) versus distance from primary jet exit plane (x) for engine/test cell	86
16. Test configuration for the study of cell heating	87
17. Variable Area Ejector test configuration	88
18. Cell and wall pressures compared to experimental data and integral method results	88
19. Base flow test case geometry and computed Mach number contours	89
20. Base flow test case computed versus experimental base static to total pressure ratio (P/P_0)	89
21. Hypersonic inlet test hardware (P-2 inlet is shown. P-12 is slightly different)	90
22. Comparison of PARC2D results to data at Mach 7.4	91
23. Boundary layer growth in a hypersonic inlet	93
24. Geometry for the convergent-divergent nozzle	94
25. Surface static pressure for the convergent-divergent nozzle	95
26. Subsonic diffuser geometry	96
27. Surface static pressure for the subsonic diffuser	96

FIGURE	Page
28. Geometry and basic formulation for flat plate calibration case	97
29. Supersonic nozzle flow basic formulation	98
30. Near field results for supersonic nozzle	99
31. Far field results for supersonic nozzle	100
32. Rearward facing step basic formulation	101
33. Velocity profiles downstream of rearward facing step	102
34. Gridding used in the calibration of the PARC2D program for steady state performance predictions (every other J indice shown)	103
35. Grid patches (Diffuser 1 shown, Diffuser 2 similar)	104
36. Mixing patch grid resolution	104
37. Boundary conditions applied over the computational domain	105
38. Started diffuser initial conditions	106
39. Unstarted diffuser initial conditions	107
40. Mach number contours for the cell volume parametric study	108
41. Velocity vectors for the cell volume parametric study	109
42. Cell pressure as a function of iteration count for the cell volume parametric study	110
43. Flow field in the cell volume for the cell volume parametric study	111

FIGURE	Page
44. 240 x 100 grid for the diffuser grid- ding parametric study	112
45. Results from using the 240 x 100 grid in the $L/D = 4$ diffuser	113
46. Cell pressure as a function of itera- tion count comparison for the initial cell volume pressure parametric study	115
47. Cell pressure as a function of itera- tion count started with the unstated diffuser intial conditions	116
48. Flow solution for COFMIX = 0.09	117
49. Flow solution for COFMIX = 0.13	118
50. Cell pressure as a function of itera- tion count for the turbulence mixing coefficient parametric study	119
51. Cell pressure as a function of itera- tion count for Case 1	120
52. Flow solution for Case 1	121
53. Diffuser wall static pressure compari- son for Case 1	122
54. Cell pressure as a function of itera- tion count for Case 2	123
55. Flow solution for Case 2	124
56. Nozzle flow solution for Case 2	125
57. Diffuser wall static pressure compari- son for Case 2	126
58. Cell pressure as a function of itera- tion count for Case 2 with COFMIX = 0.13	127
59. Cell pressure as a function of itera- tion count for Case 3	128

FIGURE	Page
60. Flow solution for Case 3	129
61. Diffuser wall static pressure comparison for Case 3	130
62. Cell pressure as a function of iteration count for Case 4	131
63. Flow solution for Case 4	132
64. Diffuser wall static pressure comparison for Case 4	133
65. Cell pressure as a function of iteration count for Case 5	134
66. Flow solution for Case 5	135
67. Nozzle flow solution for Case 5	136
68. Cell pressure as a function of iteration count for Case 1 with the modified turbulence model	137
69. Flow solution for Case 1 with the modified turbulence model	138
70. Diffuser wall static pressure comparison for Case 1 with the modified turbulence model	139
71. Cell pressure as a function of iteration count for Case 3 with the modified turbulence model	140
72. Flow solution for Case 3 with the modified turbulence model	141
73. Nozzle flow solution for Case 3 with the modified turbulence model	142
74. Diffuser wall static pressure comparison for Case 3 with the modified turbulence model	143

FIGURE	Page
75. Cell pressure as a function of iteration count for Case 4 with the modified turbulence model	144
76. Flow solution for Case 4 with the modified turbulence model	145
77. Diffuser wall static pressure comparison for Case 4 with the modified turbulence model	146

LIST OF SYMBOLS

A_d	Diffuser cross section area
A_e	Nozzle exit area
A_t	Nozzle throat area
COFMIX	Turbulent mixing coefficient in the algebraic turbulence model
D	Diffuser diameter
L	Diffuser length
$\dot{m}$	Primary mass flow exiting the nozzle
$\dot{m}_{cell}$	Secondary mass flow exiting the cell volume
$\dot{m}_{ex}$	Diffuser exit mass flow rate
Mn	Mach number
P_{cell}	Pressure in the cell volume
P_{ex}	Diffuser exit pressure
P_t	Total pressure of the primary fluid
o_R	Degree Rankine
T_t	Primary stream total temperature
T_{cell}	Cell volume total temperature
R	Primary stream gas constant
R_{cell}	Cell volume gas constant

1 INTRODUCTION

Diffusers, also commonly known as ejectors, jet pumps, or ejector-diffusers, play an often unseen role in many aspects of everyday life. Their ability to move large quantities of vapor and/or gas at low pressure with low cost and a minimum of moving parts has made the diffuser a favorite of the chemical industry for many years [1-2].¹ The aerospace industry has used diffusers to augment the net thrust of propulsion devices and to enhance the aircraft lift-drag characteristics by careful placement of a diffuser downstream of the engine exhaust outlet [3-4]. Closer to home, for households depending on a well as a source for water, the jet pump is a common method of transferring the water from the well to the house at a usable level of pressure and flow rate [5].

Diffusers are also commonly used in the ground testing of turbine engines and rocket motors. The Engine Test Facility (ETF) at the Arnold Engineering Development Center (AEDC) routinely tests a wide variety of propulsion systems at simulated altitude conditions. A sketch of a typical turbine engine test installation is shown in Figure 1 (the figures are presented in Appendix A). A typical rocket motor installation is sketched in Figure 2. By capturing the exhaust products from a propulsion system

¹ Numbers in brackets refer to similarly numbered references in the bibliography.

and converting the flow's kinetic energy to pressure rise, the diffuser either directly produces the simulated altitude pressure in the test cell or enhances the performance capabilities of an exhaust plant. The pressure rise provided by the diffuser is a direct function of the kinetic energy of the driving fluid stream and the amount of secondary flow captured from the test cell [6].

Testing of propulsion systems in ground test facilities is becoming more stringent, both in terms of test simulation requirements and in controlling cost. Proper sizing of the diffuser system to match the operating characteristics of the test article is vitally important [6]. In many cases, testing requirements dictate that a diffuser system be used that can increase the maximum altitude pressure simulation capabilities of the facility. The destructive nature of many tests also place a high level of importance on the proper sizing of the diffuser system. For example, a solid rocket motor test can not be repeated since the motor can be fired only once.

The physical nature of the flow in a diffuser can be extremely complex and a purely theoretical description of the flow was generally not feasible 20 years ago [6]. Several one dimensional, semiempirical methods have been developed for designing diffusers for various applications

[7-9]. Because of the simplifying assumptions needed to keep the analysis one dimensional, however, the designer does not know quantitatively important aspects of the flow in the diffuser (such as heat transfer rates and wall pressure loadings) that will permit the proper design of all aspects of the diffuser system. In recent years, Computational Fluid Dynamics (CFD) has shown promise in providing the diffuser designer with an analysis tool which will permit a detailed knowledge of the diffuser operation, helping to ensure that the diffuser is properly sized to match the given test requirements. As with any analysis tool, however, the level of accuracy of the computational methodologies must be known before the tool can be intelligently applied.

To address this issue, a calibration effort of the axisymmetrical version of the CFD program PARC2D for diffuser systems similar to those found in turbine engine and rocket motor testing has been accomplished. The influence of such parameters as grid spacing, turbulence levels, initial conditions, and boundary conditions has been studied. The results have been compared to data taken from the available literature. This thesis is organized such that diffuser operation is discussed in the following section. The test data used in the calibration effort is

considered in Section 1.2, and the PARC2D program is briefly discussed in Section 1.3. A complete statement of the problem is given in Section 1.4. In Chapter 2, a review of the literature discussing CFD program calibration and, in particular, examples of PARC2D calibration are presented. The calibration of PARC2D to the diffuser data is discussed in Chapter 3. Lastly, some conclusions and recommendations for future work are discussed in Chapters 4 and 5, respectively.

1.1 BACKGROUND

Diffuser systems are distinguished by their simple mechanical construction derived from a simple working principle which consists of transferring momentum from a high velocity jet of fluid (the primary fluid) to a volume of fluid at rest (the secondary fluid) through a microscopic energy exchange driven by mixing. This mixing causes some of the secondary fluid to become entrained with the primary fluid, resulting in an increase in pressure and temperature. Pertinent parameters for diffuser systems are shown in Figure 3. When discussing diffuser performance, a thorough understanding of four ratios and their significance is mandatory. These ratios are:

1. The driving pressure ratio, defined as the total pressure of the motive fluid divided by the diffuser exit pressure (P_t/P_{ex}).
2. The nozzle pressure ratio, defined as the total pressure of the motive fluid divided by the test cell pressure (P_t/P_{cell}).
3. The diffuser rise ratio, defined as the diffuser exit pressure divided by the test cell pressure (P_{ex}/P_{cell}).
4. The mass flow capture ratio, defined as the mass flow rate of secondary fluid entrained divided by the mass flow rate of the primary fluid ($\dot{m}_{cell}/\dot{m}$).

Two typical methods of graphically presenting diffuser performance are shown in Figure 4. In Figure 4a, the reciprocal of the nozzle pressure ratio is plotted versus the reciprocal of the driving pressure ratio; in Figure 4b, the reciprocal of the diffuser rise ratio is plotted versus the driving pressure ratio [6,8]. It should be readily apparent from the figure that the characteristic curves can be produced by either varying the parameters P_t or P_{ex} . In subsequent discussion, the primary fluid total pressure is the variable and the diffuser exit pressure is held constant.

Point A corresponds to the minimum driving pressure ratio and both the nozzle and the diffuser are defined to be "unstarted". The flow in the system at this operating condition is illustrated in Figure 5 (1). In this condition, the flow from the nozzle does not expand to the nozzle or diffuser walls but forms a core in the diffuser for some distance downstream of the nozzle. Some pumping is provided by the jet due to entrainment of the secondary fluid, but the diffuser rise ratio is only slightly greater than unity.

For converging/diverging nozzles, as the driving pressure ratio is increased by increasing P_t , the diffuser performance follows along the curve to Point B. At this point, the nozzle pressure ratio is high enough to cause the nozzle to flow full. The diffuser, however, still remains unstarted, as shown in Figure 5 (2). In this region, the test cell pressure decreases faster than P_t increases, causing the slope of the curve to increase. From Points B to C, the slope is nearly vertical, indicating that for slight increases in primary total pressure, test cell pressure decreases quite rapidly.

Point C is defined as the "started" condition and represents the maximum diffuser rise ratio obtainable for a given diffuser system. The supersonic flow from the nozzle

is expanded out to the diffuser wall, effectively decoupling the test cell pressure from the diffuser exit pressure. The flow field at the started condition is also shown in Figure 5 (3). Any increases in P_t will not decrease P_{cell} , but will in fact increase the cell pressure. Note the increase in the ordinate in Figure 4b as the total pressure, P_t , increases beyond Point C.

The effect of increasing secondary fluid entrainment on the system is shown in Figure 6. As the ratio of secondary fluid to primary fluid increases, the location of Point C steadily rises, indicating that if a secondary fluid flow source is present, a higher cell pressure will be obtained than if there was no secondary fluid flow. This aspect of diffuser performance will not be discussed further herein, and the reader is referenced to [6] for additional details on the subject.

1.2 CALIBRATION DATA

The data used for the calibration of the CFD program were taken out of [8]. This National Aeronautics and Space Administration (NASA) Technical Note discusses the effects of various dimensions on the performance of a small axisymmetrical diffuser system using high pressure nitrogen as the primary fluid. The test apparatus is

sketched in Figure 7 (note that the convergent section of the nozzle in the figure did not agree with the description given in the text of [8]). The diffuser system consisted of a primary chamber, a primary convergent/divergent nozzle, a cylindrical test cell, and a diffuser tube.

The primary chamber was a 1.00 inch diameter tube upstream of the nozzle throat that contained a flow straightening screen. This section provided a uniform flow profile to the nozzle. The total pressure of the primary fluid was measured in this chamber.

The primary nozzle consisted of a thirty degree half-angle convergent section, a 0.25 inch throat with a 0.5 inch contour radius, and a 15 degree half-angle divergent section. The length of the nozzle from the throat to the nozzle exit was 1 inch. The exit diameter was 0.75 inches. The area ratio of the nozzle was limited to 9 to avoid condensation of the primary fluid in the nozzle.

The cell volume was 2 inches in diameter and had a length of 1.25 inches. The cell volume was positioned such that the front wall of the cell was in the same vertical plane as the nozzle throat. The back wall of the cell was 0.25 inches downstream of the nozzle exit. Two

static pressure probes were located on the walls of the cell, one on the front wall and one on the top wall, to record the volume pressure.

Four diffusers were provided for the study. The inside tube diameters were 1.00, 1.25, 1.50, and 1.75 inches. The tube lengths were initially 12 inches, with the tubes being shortened to study the effects of the length to diameter ratio (L/D). Each tube had thirty static pressure taps located symmetrically along the length of the diffuser.

Selected data from [8] is presented in Figures 8, 9, and 10. The measured inverse of the diffuser rise ratio, $P_{\text{cell}}/P_{\text{ex}}$, is plotted against P_t/P_{ex} in Figure 8a for diffuser diameters varying from 1.00 inch to 1.75 inches, and the effect of L/D is plotted in Figure 8b for a constant diffuser diameter of 1.5 inches. Wall static pressures at various driving pressures for the 1.00 inch diameter diffuser with a L/D of 12 are shown in Figure 9. The wall static pressures for the 1.5 inch diameter diffuser with a L/D of 4 at various driving pressures are plotted in Figure 10. These last two plots are presented in particular because these data were used in the calibration of the PARC2D program.

The uncertainty of the data is not discussed in the reference. The method of obtaining the pressure measurements, however, is identified as mercury manometers. The pressure recordings were photographed and then deduced from the photographic negative. Although it is impossible to definitively deduce the uncertainty of the data from this information, it is expected that the uncertainty should be approximately the same as other reported pressure measurements obtained by the mercury manometer method during this time period. A review of several references published in the middle to late 1950's [10-14] indicate that the uncertainty of mercury manometer readings can vary from as low as ± 0.01 inches of mercury to as high as ± 0.1 inches of mercury. Given that 0.4893 psia is equal to 1 inch of mercury, even at the high level of uncertainty this equates to an uncertainty of only ± 0.05 psia.

1.3 COMPUTATIONAL FLUID DYNAMICS PROGRAM

The CFD program chosen for the calibration process is the PARC2D program. This program is based on a version of the Pulliam-Steger program (the NASA Ames Research Center "ARC" program) that was obtained by AEDC in 1985. The program has been extensively modified to produce a robust, general purpose Navier-Stokes flow solver. The program is

based upon a variant of the Beam and Warming implicit approximate factorization algorithm applied to the compressible Navier-Stokes equations expressed in curvilinear coordinates [15]. The mathematics of the PARC2D program are discussed in Appendix B.

Previous reported applications of the PARC2D program to diffuser problems has been somewhat limited. The use of the program to qualitatively study the interaction of a turbojet engine, test cell and diffuser at near sea-level altitude conditions is discussed in [15], but no quantitative work has been performed to date. Validation of the PARC2D program for several types of classical flow problems was accomplished during 1988 [16], but the comparison of PARC2D calculations to actual data for diffuser flows has not previously been reported.

The results presented in this thesis were obtained by exercising the PARC2D program on a Cray X MP/12 at the AEDC Central Computer Facility. Average CPU requirements for a typical case in this study were 6 minutes for every 1,000 iterations. The number of iterations required for each case varied from 40,000 for the started diffuser cases to 100,000 for the unstarted diffuser cases. Computer resource expenditures were approximately \$2,800 for each started diffuser case and \$7,000 for each unstarted

diffuser case. The problems averaged 18,000 grid points in the computational domain, requiring approximately one megaword of memory. The use of nitrogen as the primary fluid in [8] was accounted for in the PARC2D program by using the proper values for the Sutherland viscosity law temperature and the ratio of specific heats in the PARC program input file.

1.4 STATEMENT OF PROBLEM

The calibration of the PARC2D program will be broken into three sections. The first part will be a parametric study of the relative influence of perturbations to the solution technique on the calculated flow field. Variations in the cell volume, the diffuser grid resolution, the initial conditions used to start the solution, and the level of turbulent mixing will be made and the results discussed.

The second section will be the actual calibration of the PARC2D program to data. The data, taken from [8], will be for an axisymmetrical diffuser system employing a nozzle with a nozzle exit to throat area ratio (A_e/A_t) of 9 and two diffusers; the first diffuser will provide a diffuser area to throat area ratio (A_d/A_t) of 36 with a

L/D of 4 (Diffuser 1), while the second diffuser will provide an A_d/A_t of 16 and a L/D of 12 (Diffuser 2). Five operating conditions will be considered.

The calibration process will first tune the PARC2D results to match the diffuser rise ratio for the started case with Diffuser 1 via the turbulence mixing coefficient (see Appendix B). The remaining four cases will then be analyzed with the program "as is", with the results being compared to both the diffuser rise ratio and the wall static pressures down the length of the diffuser. The study will also consider a modification to the free boundary turbulence model that adjusts the strength of the turbulent mixing based on the local flow Mach number. Results will be presented for three of the cases as above. The results of these investigations are presented in Chapter 3.

2 LITERATURE SURVEY

The calibration and validation of CFD programs has been a very active research topic in the recent past. As more and more dependence is placed on CFD calculations, the sources of error in the computational algorithms must be fully understood. It is of extreme importance that CFD programs be tested by comparing them to appropriate experiments to both ensure their validity and to define their applicable range. Several examples of the calibration of the PARC2D program exist in recent literature, including such flow calculations as supersonic inlets, axisymmetric and two-dimensional nozzles, diffusers, and exhaust gas management systems. The following literature survey is intended to acknowledge earlier calibration efforts on the PARC2D program and the authors. It is not intended to completely cover all of the available literature relating to CFD program calibration.

The terminology used in the present discussion will be based on the definitions for calibration and validation as given in [17]. This work has received considerable attention in the literature ([18] and [19], for example), and was based on the work of a NASA ad hoc Committee on Validation. The pertinent definitions are:

CFD program validation: Validation provides detailed surface and flow field comparisons with experimental data. The purpose of the comparison is to verify the program's ability to correctly model the critical physics of the flow. It is required that the accuracy and limitations of both the experimental data and the program be thoroughly known and understood over a range of specified parameters.

CFD program calibration: Calibration provides a comparison of a program's ability to accurately predict certain aspects of the flow field by comparing the program output to experimental data obtained from geometries that are similar to the one under consideration. It is important to ensure that only the parameters of interest are acceptably predicted and not that all aspects of the flow are correctly modeled.

As a general observation, the concept of a validated PARC2D program is presently an unachievable goal. The current state-of-the-art in many aspects of CFD dictates that proper modeling of the critical flow physics can not occur. The most obvious example of this is in the modeling of turbulence. Although just one of many sources of error in CFD [20], turbulence modeling as it currently stands, with empirically based models, has a long way to go before an accurate reproduction of the flow physics is

obtained [21].

The first recorded example of the calibration of the PARC2D program was provided by Huddleston, Cooper, and Phares [22]. This effort used the PARC2D program to study the effects of test cell exhaust gas recirculation on the surface pressure distributions occurring on a high-bypass turbofan engine. Two geometries were considered, the first being a subscale test configuration and the second being an actual turbine engine installation in an ETF test cell. The subscale test case geometry is shown in Figure 11. The PARC2D calculations matched the test data to within approximately ± 5 percent for both the center pipe surface pressures and the cell wall pressures. The comparison plots are shown in Figure 12 and 13. The subscale case provided a calibration of the PARC2D program and indicated that further effort was warranted. The actual turbine engine test case had a geometry similar to that shown in Figure 1. Measured and computed fan cowl surface pressures are compared in Figure 14. Likewise, core plug surface pressures are compared in Figure 15. Although the computed values were within ± 25 percent (with a few exceptions), the exact location of the predicted peak locations were somewhat shifted from those recorded. The

discrepancy was considered to be the result of both the axial and normal spacing of the gridding, and further investigation was recommended.

Another example of using the PARC2D program in a turbine engine testing environment was provided by Phares, Cooper, Swafford, and Jones [15]. This reference discusses several applications of the PARC2D program to problems occurring in propulsion testing. Pertinent applications included a study of test cell heating due to diffuser and engine interaction, the prediction of diffuser wall pressures in a variable area ejector, and the prediction of base pressures for a centered propulsive jet. The study of the test cell heating due to diffuser and engine interaction was a qualitative comparison of PARC2D calculations to experimentally obtained Thermovision camera pictures. The geometry of the hardware is shown in Figure 16. For a given test cell pressure, the PARC2D program was able to qualitatively predict the presence of exhaust gas recirculation from the diffuser. The geometry of the variable area ejector case is shown in Figure 17. The results, which compared PARC2D calculations to both test data and an integral analysis method [23], are plotted in Figure 18. The PARC2D program calculations matched diffuser wall static pressure to

within ± 5 percent if the diffuser wall boundary condition was defined as a constant specified temperature. If the wall was treated as adiabatic, errors as large as 25 percent were generated. In addition, the centered propulsive jet case was used to determine the capability of the PARC2D program to determine base pressures. The geometry of the case is shown in Figure 19. Comparison of the PARC2D results to test data is shown in Figure 20. As shown, the data compares reasonably well with the data, with an error of approximately 10 percent.

Hunter and Kent [24] document the use of the PARC2D program to analyze two hypersonic inlets at Mach 7.4. The geometry for the two inlets, which were designated in [24] as P-2 and P-12 inlets, is presented in Figure 21. Calibration of the PARC2D program was obtained by comparing program output to centerbody static pressures and cowl static pressures. These comparisons are shown in Figure 22. As shown, the PARC2D program predicted centerbody and cowl static pressure distributions for the P-2 inlet to within approximately 15 percent. Centerbody static pressure distributions for the P-12 inlet indicated a discrepancy between the program results and the data of as much as 30 percent. No discussion was included in [24] as to why the differences might have occurred. The cowl static

pressure predictions were within 10 percent of the data.

The work performed by Semmes, Arbiter and Dyer [25] is an excellent example of a calibration effort applied to the PARC2D program. Three different configurations were considered for the calibration effort, including a two-dimensional hypersonic inlet (same data set generated by the P-2 inlet in [24]), an axisymmetric convergent-divergent nozzle, and an axisymmetric subsonic diffuser. For the two-dimensional hypersonic inlet, a considerably broader amount of data was considered in [25] than was in [24]. For example, the boundary layer growth along the centerbody and cowl predictions compared to data are shown in Figure 23. The PARC2D boundary layer thickness predictions were as much as 50 percent larger than the test data. The geometry for the axisymmetric convergent-divergent nozzle is given in Figure 24. Surface static pressure along the top wall is shown in Figure 25. The PARC2D program predicted the wall static pressures to within 10 percent both upstream and downstream of the nozzle throat. In the throat region the pressure predictions were off by as much as 50 percent. The computational grid used for the subsonic diffuser case is presented in Figure 26. As above, the surface static pressure distribution is plotted in Figure 27. The errors

in the PARC2D predictions were near 0 percent at the inlet of the nozzle and increased to approximately 5 percent at the exit of the diffuser.

The final selection that was reviewed was the work reported by Cooper, Garrard, and Phares [16]. The document reported on work performed to validate the PARC2D program for three fundamental flows typical of turbine engine and rocket motor testing at the AEDC. Laminar and turbulent flow over a flat plate was investigated, as was a supersonic, axisymmetrical jet issuing from a Mach 2.22 nozzle into quiescent air and subsonic flow over a rearward facing step. The geometry applied in the flat plate analysis is shown in Figure 28. Numerous comparisons to data were made for both laminar and turbulent flow with various inflow and outflow Reynold's numbers, various boundary conditions treating the plate as either adiabatic or specified temperature, various inflow Mach numbers, and various grids. The discrepancies between the PARC2D calculations and the calibration data were limited to 5 to 10 percent, even for such parameters as skin friction and Stanton number. The supersonic nozzle geometry is presented in Figure 29. The test case was a Mach 2.22 flow exhausting into quiescent air with no thermal gradient. The near field results, shown in Figure 30, indicate that

the PARC2D program matched the spread rate of the turbulent jet. Far field results, shown in Figure 31, however, show that the PARC2D program underpredicted the spread rate by approximately 40 percent at $X/R_{ne} = 140$. The geometry of the rearward facing step is presented in Figure 32. The PARC2D program results are compared to data in Figure 33. The PARC2D program predictions matched velocity profiles to within ± 10 percent everywhere except in the separated region just downstream of the step. In the separated region the difference between the predictions and data were as high as 30 percent.

3 CALIBRATION OF THE PARC2D PROGRAM

This section will detail the work performed to calibrate the PARC2D program for predicting steady state diffuser performance parameters. The work will be introduced by a discussion of several general characteristics of the approach taken for the effort. The results obtained during a parametric study will then be discussed. The comparison of the PARC2D results to the calibration data will then be made, and finally, the results obtained after a modification to the algebraic turbulence model will be discussed.

3.1 GENERAL

Several aspects of the calibration study have been consistently applied throughout. Rather than duplicate the presentation of these aspects in each section, they will be discussed below. The parameters of interest include gridding, boundary conditions, initial conditions, and convergence criteria.

Since the calibration effort was limited to only two axisymmetrical nozzle/diffuser configurations, it was necessary to provide only two different grids. The grids, developed using the techniques developed in [26], are

shown in Figure 34. Each grid was constructed of 18,000 grid points along 180 vertical lines (referred to as the I indices) and 100 horizontal lines (referred to as the J indices). The first grid provided the computational domain for the 1.5 inch diameter, 6 inch long diffuser ($L/D = 4$), and the second grid modeled the 1.0 inch diameter, 12 inch long diffuser ($L/D = 12$). The nozzle and cell region was identical in each grid. The grids were built by breaking the nozzle and diffuser system into four patches, where the first patch represented the nozzle, the second patch represented the cell volume, the third patch represented the mixing zone between the nozzle exit and the diffuser, and the fourth patch represented the diffuser. The area above the subsonic section of the nozzle was not used in the study and was included only to simplify the gridding. The grid patches are identified in Figure 35.

The nozzle patch was constructed of I indices 1 to 80 and J indices 1 to 40. The nozzle throat was located at an I index of 40. The grid spacing along the nozzle wall was exponentially packed with an initial spacing of 0.0001 inches. The cell volume patch was likewise constructed of I indices 1 to 80, and the J indices varied from 41 to 100. The cell volume between I indices 1 to 40 was not

used in the calibration effort, since the actual test hardware placed the cell front wall in the same vertical plane as the nozzle throat. Along I index 40, the J indices were equally spaced. At the nozzle exit, however, the packing was directed towards the nozzle lip with a spacing of 0.0001 inches.

The mixing zone, containing I indices 80 to 90 and J indices 1 to 100, represented the transition zone between the nozzle and cell volume patches and the diffuser patch. It was here that the grid had to smoothly transition into the geometry of the particular diffuser. This was accomplished by taking the J index distribution given on I index 80 and providing a J index distribution on I index 90 as shown in Figure 36. The I indices were equally spaced in the 0.25 inches between nozzle exit and the diffuser inlet. In the only area that deviates from the actual test geometry, the outer cell wall boundary was reduced to the diffuser wall diameter between I indices 80 and 90. This was accomplished over the 0.25 gap, giving a finite slope to the wall rather than being vertical as in the actual hardware. Since the wall was nearly vertical and was located completely out of the main flow, no significant effects were expected from this modification.

The diffuser patch was constructed of I indices 90 to 180 and J indices 1 to 100. The J indices were packed exponentially against the diffuser wall with the first index 0.0001 inches off the wall. The I indices were packed towards the nozzle exit with an initial spacing of 0.025 inches. The physical coordinates of the diffuser varied with the diffuser under consideration. A variation of the 1.5 inch diameter diffuser was developed for this study, and is discussed in the parametric study section. This grid had the number of I indices increased from 180 to 240 while keeping the same system geometry. The extra indices were placed in the diffuser to increase the grid packing.

The axisymmetrical grids described in this section were intended to provide a grid that would furnish an adequate computational domain for the problem solution, but also not require large amounts of computer memory and processing time. These grids were a good compromise. It should be noted that based on the definitions of validation and calibration discussed in Chapter 2, had this been a validation effort, gridding considerations would have been of more importance. Since this was a calibration effort, however, it was deemed necessary to provide grids that would be similar to those used in typical diffuser

applications.

Boundary conditions applied on the computational domain are shown in Figure 37. All walls were treated as no-slip and adiabatic with the exception of the bottom and front wall of the cell volume which were considered as slip surfaces. Because of the low flow velocity expected in these regions, it was judged that resolving a boundary layer along the wall would not contribute to the solution and would in fact degrade the solution by requiring significant computer processing time. The exit of the diffuser provided the outflow boundary. This boundary condition was provided by using the diffuser exit pressure from [8], which varied from 14.3 psia to 14.4 psia. For this study, the exit pressure was held a constant 14.3 psia. Flow into the system was controlled on I index 1, J indices 1 to 40. The total pressure of the nitrogen flow was applied there, the value of which depended on the condition being studied. The axis of symmetry for the diffuser system was defined along J index 1.

The initial conditions applied over the computational domain were varied depending on the expected performance from the diffuser. For data points with the diffuser started, which would fall to the left of Point C on the diffuser performance map (see Figure 5), the initial con-

ditions were defined as shown in Figure 38. The flow properties inside the nozzle were based on the total pressure required for the data point of interest and the nozzle flow area at the I index under consideration. One dimensional, isentropic flow was assumed, with subsonic flow defined upstream of the nozzle throat and supersonic flow defined downstream of the nozzle throat. Downstream of the nozzle exit, the flow continued down the diffuser with the nozzle wall boundary ($J = 40$) extended to the diffuser exit to provide an artificial boundary for the flow. The cell volume and the remaining grid points in the diffuser were given properties corresponding to very low velocity flow at a pressure near the desired answer. The effect of this pressure level on the final solution was investigated and will be reported in the next section.

The initial conditions for data points operating with an unstarted diffuser are shown in Figure 39. The flow in the nozzle was exactly identical to the started case. Outside of the nozzle, however, the entire grid was set to the pressure and flow velocity as in the cell region of the preceding case. The initial conditions were set this way due to difficulties that were present when the initial conditions shown in Figure 38 were used on some of the unstarted cases. With the diffuser filled with supersonic

flow, the solution resulted in the diffuser starting. Before the unstarted pressure could be obtained in the cell volume, a shock would have to travel up the diffuser. Waiting for the shock to reach the entrance to the diffuser was time consuming and required a large number of iterations on the computer (greater than 20,000 iterations). By starting the supersonic flow from the nozzle exit, the convergence problem was accelerated without impacting the final results.

The criteria used for convergence in this study was based on two variables. First, the sum of the root mean square of the individual residuals of mass flux, momentum fluxes, and energy flux throughout the grid was required to be less than 0.000001. Secondly, the flow field could not change when viewed approximately every 2000 iterations. Mach number contours were plotted and used for the comparison, which was qualitative in nature. For the started diffuser cases, convergence was obtained within approximately 50,000 iterations. The supersonic region of the flow field reached convergence considerably earlier (approximately 15,000 iterations), but the subsonic flow field in the cell volume and against the diffuser wall required the additional iterations in order to reach steady state. The unstarted diffuser data points were to

be treated likewise, but because of the unsteady nature of the flow, no converged solutions were obtained for two of the three cases considered. This aspect of the study will be discussed in greater detail in the Results section.

3.2 PARAMETRIC STUDY

To begin the calibration effort on the PARC2D program, a series of parametric studies were conducted. These studies were used to provide an understanding of the sensitivity of the PARC2D program to certain changes in the solution technique. A detailed discussion of the results of each solution is given below. The flow conditions used in the studies were a total driving pressure of 1000 psia and a total temperature of 580 °R. The high total pressure was selected (the maximum driving pressure used in [8] was 629 psia) in order to ensure that the diffuser system remained started during the parametric changes. There was no secondary flow. Since there were no data collected at this pressure in [8], no comparison to data can be made in this section. All calculations were made with the diffuser having an L/D of 4.

CELL VOLUME EFFECTS

Three solutions were used to investigate the effect of the test cell volume on the final cell pressure. The motivation for conducting this part of the investigation was to see if a computational volume less than all of the cell volume could be used during the aerodynamic calibration. The smaller the cell volume, the fewer grid points that would be part of the solution. It was judged that if fewer grid points were used in the slow converging subsonic flow region, a direct impact on the computer CPU time required to obtain a converged solution could be made.

The first case, which provided the baseline solution, used a cell volume equivalent to that in [8], where the vertical front wall of the cell was in the same axial plane as the nozzle throat. The second case removed half of the cell volume by moving the front wall up half way between the nozzle throat and nozzle exit planes. The third case reduced the cell volume to a minimum by moving the front wall up to within 0.05 inches from the nozzle exit plane. All other parameters were held constant in all three cases to ensure that any variations observed were indeed due to the changing cell volume. The study

looked in particular for any variation in cell pressure that would have indicated a change in the solution from the baseline case.

The results from the three cases are presented in Figures 40 and 41. Mach number contours varying from 0.5 to 5.0 in 0.5 increments are shown in Figure 40 and velocity vectors are plotted in Figure 41 (the direction of the vector indicates flow direction, and the length of the vector is proportional to the flow velocity). There was visually no qualitative difference between the solutions in the diffuser. The predominant flow feature computed was a barrel shock generated by the nozzle exit flow turning near the diffuser wall. There also was a region of separated flow approximately 2.5 L/D's down the diffuser. Given that the ratio between the ambient diffuser exit pressure and the diffuser flow static pressure was as high as seven, the separated flow region was expected.

The most significant difference between the three cases was in the final cell pressure that was obtained. The cell static pressure, which was located on the cell top wall in the same plane as the nozzle exit (where the grid starts reducing the diameter to match the diffuser), is plotted in Figure 42 for all three cases as a function of iteration count. Other cell pressure measurement loca-

tions showed the same pressure value after approximately 1000 iterations and will not be shown. After the first 5000 iterations, where the initial conditions were being washed out of the system, the cell pressures quickly converge to about their final level. For the remainder of the iterations, however, it was a slow process to obtain the final values. The final pressures clearly show the effect of varying cell volume. The first two cases, where the full cell volume was used and only half of the cell volume was used, converged to approximately the same answer. There was a slight difference, with the half volume case being 3 percent higher (0.02 psia) than the full volume case. The third case, however, converged to a cell pressure approximately 10 percent lower than the other two cases. The reason for this can best be explained by looking at Figure 43. Computed velocity vectors in the test cell region for all three cases are plotted in the figure. For the full volume and half volume cases, the computed flow features in the cell volume were similar in that the recirculating flow coming off of the jet boundary was able to set up one or more vortices in the cell with low velocity. In the third case, however, the recirculating flow vortex did not have a chance to dissipate into a

relatively large cell volume. This caused the flow velocity to be maintained, helping to generate a lower pressure in the cell.

It was concluded from this parametric study that the cell volume could be safely reduced to approximately half of the volume used in [8]. Reducing the volume any further could potentially introduce unacceptable errors in the solution. However, since a 50 percent reduction in cell volume did not reduce the number of iterations required to obtain a converged solution, it was decided that the remaining work would use the full cell volume to ensure complete compatibility with the work reported in [8].

DIFFUSER GRIDGING EFFECTS

The second parametric study conducted looked at the effect of increasing the resolution of the grid in the diffuser. To accomplish this, a grid was generated that increased the number of I indices from 180 to 240, with the additional 60 indices being located in the diffuser. The geometry of the system did not change. Only the spacing of the I indices in the diffuser was modified. The intent of the study was to determine if the higher grid resolution in the diffuser would result in a different

computed flow field. The grid used for this study is shown in Figure 44. Since the grid spacing at the inlet of the diffuser did not change, the additional indices resulted in an axial distance between indices at the exit of the diffuser of only 0.059 inches, versus 0.134 inches for the other grid.

Mach number contours and velocity vectors are plotted for the solution in Figure 45a and 45b, respectively. Referring back to the full grid results plotted in Figures 40 and 41, there was no apparent differences between the two grids. The extra gridding did give better visual resolution of the shock waves reflecting off the centerline and the wall downstream of the barrel shock. This was most apparent in the velocity vector plots; the Mach number plots indicated that the original grid provided almost the same flow structure as the new grid. The region of separated flow along the diffuser wall was also still present and was of the same appreciable size as before. A more direct comparison between the two cases is presented in Figure 45c by plotting the diffuser wall static pressures down the length of the diffuser. Other than the shock location being shifted approximately 0.25 inches to the left for the 240 x 100 grid solution, there was no significant differences between the two solutions.

Besides not providing any significant benefits, the higher resolution grid also carried several negative aspects. Because of the additional grid points (a 33 percent increase), more computer memory was required to exercise the PARC2D program. The 180 by 100 grid required less than 1 megaword of memory, while the 240 by 100 grid increased the memory requirement to 1.5 megaword. This additional memory resulted in a low priority being placed on computer run, adding approximately one week clock time to the solution effort. Based on these reasons, it was concluded that there were no benefits to be gained by using the 240 by 100 grid. The remaining parts of the calibration effort used the 180 by 100 grid.

INITIAL CELL PRESSURE EFFECT

In most diffuser applications of the PARC2D program, the user will not know beforehand the exact value of the cell pressure (P_{cell}) that will result. Since some value of pressure must be used in the initial conditions, it was the intent of this parametric study to investigate if using either a lower or higher pressure than the actual final answer would provide a faster convergence rate.

Two cases were considered in this study. The first case, which was the same case used above for the full cell

volume study, used an initial cell pressure of 0.5 psia which was approximately 25 percent lower than the final cell pressure. The second case used an arbitrary cell pressure equal to 2.0 psia which was higher than the final cell pressure by 65 percent. The cell pressures for both cases as a function of iteration count are plotted in Figure 46. The 0.5 psia cell pressure case converged to the final solution in approximately 20,000 iterations. It took the second case 40,000 iterations to reach its final solution. At that point, the cell pressure was approximately 10 percent higher than that obtained in the first case.

Based on these results, it can be concluded that using a cell pressure lower than the expected value can result in an improved rate of convergence in the solution. It should be noted that the comparison is incomplete, however, since the second case was a much higher percentage away from the final solution than the first case. For the remainder of this study, the cell pressure was set to within 25 percent of the final P_{cell} and the fact that the pressure is higher or lower than the final solution will not be a driving consideration.

INITIAL DIFFUSER FLOW EFFECT

The effect of the type of initial conditions specified in the diffuser were also studied. In particular, the study looked at using full isentropic flow in the diffuser versus having no flow at all in the diffuser and setting the diffuser initial conditions equal to the initial conditions specified in the cell volume. The initial conditions used in the study are shown in Figure 38 for the started diffuser case and in Figure 39 for the unstarted diffuser case.

The cell pressure as a function of iteration count for the unstarted diffuser initial conditions is plotted in Figure 47. The results obtained using the started diffuser initial conditions are presented in Figure 46 by the 0.5 psia initial cell pressure curve (the results from the initial cell pressure effect study have been utilized). It is shown that even though the full flow initial condition case should have been considerably closer to the final solution, there was no appreciable difference in the number of iterations required to achieve convergence. This was not totally unexpected, since as stated earlier, the supersonic portion of the flow field converged rather quickly. It was the subsonic portion of the flow field that required a large number of iterations. Since the

majority of the diffuser flow was supersonic, the initial conditions in the diffuser did not have a direct influence on the convergence rate.

It was concluded from this study that for regions where the flow field will be mainly supersonic, the initial conditions do not play a critical role in the overall rate of convergence. The regions of subsonic flow completely override the supersonic flow regions in controlling the rate of convergence of the flow solution. Based on these results, either set of initial conditions was used in the remaining work efforts.

TURBULENCE MIXING COEFFICIENT EFFECT

The turbulence mixing coefficient (COFMIX) controls the level of turbulence introduced to the flow by the algebraic turbulence model (see Appendix B). As such, its value directly controls the flow solution that is obtained. For this parametric study, the value of COFMIX was changed from a nominal value of 0.11 to 0.09 and then to 0.13. The solution with COFMIX equal to 0.11 is shown in Figures 40 and 41, the full cell volume case. Mach number contours and velocity vectors for the solution with COFMIX equal to 0.09 are shown in Figure 48. The solution with COFMIX equal to 0.13 is shown in Figure 49. As in

the previous parametric studies, the major difference between the three solutions was in the final cell pressure value obtained. The cell pressure for each case as a function of iteration count is shown in Figure 50. As can be seen, the higher COFMIX was set, the lower the cell pressure. A ± 20 percent change in COFMIX resulted in an approximately ± 30 percent change in cell pressure. This was expected, since a higher value of COFMIX directly increased the level of mixing in the jet boundary. A higher mixing level resulted in a higher level of mass entrainment by the jet, thereby reducing the amount of mass in the cell volume. A lower amount of mass equated to a lower cell pressure. The opposite occurred for a low value of COFMIX.

Unlike the other parts of the parametric studies, there were no quantitative conclusions to be drawn from this section. It was shown, however, that the turbulence mixing coefficient can be used to adjust the overall performance predictions for a given diffuser system. This fact will be of great importance in the aerodynamic calibration discussions which follow.

3.3 AERODYNAMIC CALIBRATION

To determine the performance of the PARC2D program for predicting steady state diffuser performance parameters, comparisons to data were made for two diffuser configurations operating in the three regimes discussed in Chapter 1. This section will detail the methodology used and discuss the results obtained for each case. The first configuration considered used the diffuser having a L/D of 4. The second configuration used the diffuser having a L/D of 12. For each diffuser, the PARC2D program was calibrated when operating in the started mode using the turbulent mixing coefficient to obtain the correct cell pressure. Then, once the required value of COFMIX was determined, the program was exercised with one or more unstarted diffuser cases.

CASE 1

The first case to be analyzed was based on the maximum pressure case for the L/D = 4 diffuser. The total pressure for the primary nitrogen flow was 629 psia. The started diffuser initial conditions as shown in Figure 38 were applied to the computational domain in order to begin the analysis.

In order to obtain the correct cell pressure of 0.43 psia, values of 0.11, 0.115, and 0.12 were tried for COFMIX. The value of 0.115 provided a cell pressure very nearly equal to the data, as is shown in Figure 51. The resulting flow field, shown by Mach number contours and velocity vectors, is given in Figure 52. There was visually not much difference between these results and those obtained in the parametric study presented in Section 3.2 other than the shock location. The shock was located farther up in the diffuser due to the lower driving pressure ratio.

A comparison between the calculated and measured wall static pressures down the diffuser length are presented in Figure 53. The calculations followed the data until just downstream of where the nozzle exit flow reached the diffuser wall. At that point, the PARC2D program overpredicted the rate of pressure increase along the wall resulting in the sudden pressure recovery seen in the figure. These results indicate that the PARC2D program did not correctly model the flow in the diffuser. Based on the discussion of normal shocks in ducts in [27] and the test data from [8], it was expected that the pressure recovery in the diffuser would result from a series of normal shocks. The computational results shown in Figures

52 and 53, however, indicated that the PARC2D program set up a single strong normal shock in order to effect the needed pressure rise. The computed flow field compared favorably with the flow field shown in Figure 5.15(a) in [27] for flows with a thin boundary layer or a low pressure ratio.

CASE 2

The second case to be considered in the study was a condition operating between the points A and B in Figure 5. The total pressure of the nitrogen primary fluid was 208 psia. At this operating pressure, the diffuser was not flowing full, and the flow was separated inside the nozzle [8]. The turbulence mixing coefficient was set to 0.115 as defined above. The solution was started with started diffuser initial conditions as shown in Figure 38.

The cell pressure, P_{cell} , as a function of iteration count is shown in Figure 54. As can be seen, P_{cell} initially indicated that the diffuser was operating in the started mode. After 20,000 iterations however, the diffuser had begun operating in the correct regime. The cell pressure was 30 percent higher than the test data value of 9.5 psia. The plots of Mach number contours and velocity vectors are shown in Figure 55. Examining the

flow in the nozzle more closely, which is presented in Figure 56, reveals that the flow did not separate from the nozzle wall until just before the nozzle exit. Since the data in [8] does not include the location of the flow separation point in the nozzle, it was not known if the computed flow field separated at the correct location. If the flow did separate too late, the cell pressure could have been driven higher due to improper nozzle exit flow conditions. Wall static pressures, both calculated and measured, are plotted in Figure 57. There was not much to be concluded from this plot since the cell pressure was considerably above the test data. There was an indication of a region of accelerating flow, however, which was not expected given the adverse pressure gradient in the diffuser.

Other values of COFMIX were investigated to see if perhaps one value would be required for started cases, another value for unstarted cases. The cell pressure as a function of iteration count for a value of COFMIX of 0.13 is plotted in Figure 58. This plot was generated by continuing the solution from above and simply increasing the mixing coefficient. It was not started from initial conditions. The higher mixing coefficient did effect a lowering of the cell pressure by approximately 0.8 psia

within 6,000 iterations. It should be mentioned, however, that the time step size had to be reduced in order to keep the flow solution stable. Given these results, the mixing coefficient was raised to 0.15. At that level, the flow solution became unstable and could not be converged. It was therefore concluded that the PARC2D program could not adequately handle this regime of the diffuser performance map.

CASE 3

This case was to be used to investigate the region lying between points B and C in Figure 5. When the diffuser operates in this regime, the nozzle flows full, but the diffuser does not. For this particular case, the total pressure of the nitrogen primary fluid was 360 psia. As with Case 2, the turbulence mixing coefficient was set to 0.115. Given the results from Case 2, which indicated that the diffuser started during the first part of the solution, the initial conditions assumed the unstated diffuser values as shown in Figure 39.

The results obtained for this case were quite interesting. It appeared that, although the PARC2D program was converging to the correct cell pressure, the solution was highly transient. The cell pressure as a function of

iteration is shown in Figure 59. A total of 100,000 iterations were required to develop the cyclical nature of the solution, and it was not determined if additional iterations would have resulted in the solution converging to a constant pressure. The version of the PARC2D program used for this case was not time accurate and as such, no time dependent conclusions can be drawn from the results. It is interesting to note, however, that [8] describes the diffuser operation in this regime as being unstable, and in fact identifies several points (including this one) as having a "buzzing" nature.

The flow solution obtained for this case is presented in Figure 60. The velocity vector plot indicates a region along the diffuser wall where the flow has reversed direction. This qualitatively agrees with AEDC testing experience where water from exhaust gas coolers downstream of the diffuser has been observed to travel up into the test cell when the diffuser is operating in the unstarted mode. It should also be noted that this phenomena was also present in Case 2, as shown in Figure 55.

The wall static pressure down the diffuser, both calculated and data, are shown in Figure 61. The PARC2D values were obtained at 100,000 iterations. Although the

test cell pressure is over 10 percent too low, the pressure recovery in the diffuser occurs as it should with the pressure rise taking the entire length of the diffuser.

CASE 4

This case was a simulation of the diffuser system operating in the started mode (to the left of Point C in Figure 5) with the diffuser having a L/D of 12. The total pressure of the nitrogen primary fluid was 460 psia, resulting in the diffuser operating to the left of Point C in Figure 5. The turbulence mixing coefficient was set to the nominal 0.115. Since the diffuser was operating in the started mode, the started diffuser initial conditions, as shown in Figure 38, were used to start the analysis.

The cell pressure as a function of iteration count is presented in Figure 62. As can be seen, the value of 0.115 for COFMIX was correct, as the calculated value for cell pressure quickly converged to the 1.55 psia given in the data. The overall flow field calculated by the PARC2D program is shown in Figure 63. In both the Mach number contour plot and the velocity vector plot, the shock structure inside the diffuser was readily apparent.

The wall static pressure down the diffuser is plotted in Figure 64 for both the calculated values and the data.

For this particular case, the PARC2D program does an excellent job of predicting the wall static pressure. Unlike the Case 1 results, the program did not overpredict the shock strength in the diffuser. The wall static pressures remained within 10 percent of the data until nearly half way down the diffuser. Beyond that, the pressures still remained within 50 percent of the data and appeared to be shifted slightly, rather than fundamentally wrong as the Case 1 results.

CASE 5

This case used the same diffuser as in Case 4, but the system was operating with a total pressure of 94 psia. This placed the operation of the diffuser between points A and B in Figure 5. The turbulence mixing coefficient was set to 0.115. The unstated diffuser initial conditions shown in Figure 39 were used to start the solution.

The cell pressure as a function of iteration count is shown in Figure 65. The cell pressure does not converge to the data, but appears to reach equilibrium at a cell pressure value approximately 60 percent higher than the test data. The flow field given by the PARC2D program is shown in Figure 66. The flow inside the nozzle was separated from the nozzle wall, as shown in Figure 67, but it was

not known if the separation point was correct since [8] did not provide the required data. Based on the discussion in Case 2, if the nozzle had separated too far downstream of the nozzle throat the cell pressure would be impacted due to incorrect nozzle exit conditions. Wall static pressures predicted by the PARC2D program were not compared to data for this case since the cell pressure results were significantly higher than the test data.

A case operating with the diffuser having a L/D of 12 between points B and C in Figure 5 was not attempted due to a lack of test data with which to compare.

3.4 MODIFICATION STUDY

One of the most troublesome aspects of the work reported in the above sections was the large amount of computer time required to get a converged solution. This was driven by the small time step size that could be taken without causing the numerical algorithms to become unstable. The number of iterations required to get to a converged solution is directly proportional to the size of the time step taken. If the time step size could be increased, the rate of convergence would likewise be accelerated.

One of the major factors controlling the time step size in the above examples was the turbulence level in the flow. The PARC2D program limits the rate of change of certain derivatives during the flow solution, and an artificially high turbulence level could cause the limits to be exceeded. The turbulence model described in Appendix B generates a value for the turbulent viscosity based on the formulation by Thomas [28] of the Baldwin and Lomax model [29], with the final value corrected by the constant turbulence mixing coefficient (COFMIX). This correction factor has been required simply because the values calculated without it were much too high. The problems that occurred above, however, indicated that the use of a constant correction might be too simplistic for the actual flow problem. The investigation was also driven by the results reported in [16], in particular by the complete failure of the turbulence model to adjust for the far field flow in the Mach 2.22 jet.

To investigate this problem, a modification to the turbulence model was made which provided a correction factor based on the local Mach number of the flow. Mach number was chosen as the independent variable since it provides a reasonable indication of the energy of the flow at a given location. For a given value of Mach number at

any particular grid point in the computational domain, the correction factor applied to the turbulent viscosity calculation was changed to:

$$\text{COFMIX} = \frac{1}{(1 + 0.3\text{Mn}^2)}$$

The variable was limited to a maximum value of 0.47. The value of the constants in the equation have been selected based on unreported work performed by W. J. Phares at the AEDC for the supersonic jet exhausting into quiescent air as reported in [16]. Although the constants provided excellent simulation results for that particular case, there is no guarantee that they will be appropriate for the geometry at hand.

To study the effect of this modification on the ability of the PARC2D program to predict steady state diffuser performance parameters, three of the five cases looked at above were reanalyzed. The cases were started from the same initial conditions as previously used to ensure that any differences in the convergence rate were directly attributable to the turbulence model modification. The three cases will now be discussed.

CASE 1

The cell pressure as a function of iteration count for this case, which used a total pressure of 630 psia in the diffuser having a L/D of 4, is plotted in Figure 68. Even though the time step size was increased from 0.15 seconds to 0.7 seconds, the final cell pressure was reached in approximately the same number of iterations. It was also noted that the constants in the new model provided a cell pressure approximately 80 percent higher than the test data. No effort was made to adjust the results.

The flow field generated by the new model is shown in Figure 69. The wall static pressures down the diffuser length are compared to data in Figure 70. There was no significant difference in these results from those obtained earlier, other than the higher cell pressure.

CASE 3

This case simulated the operation of the diffuser system with the diffuser unstarted and the nozzle flowing full. The total pressure of the nitrogen primary fluid was 360 psia. The cell pressure as a function of iteration count is plotted in Figure 71. As with the previous run with this case, 100,000 iterations were made. Cell pressure was overpredicted by 60 percent. Unlike the ear-

lier run, the solution behaved more like a solution in the region on the diffuser performance map lying between points A and B. The solution also showed a slight cyclic trend, which was expected, but not to the extent that the previous solution had predicted.

The flow field generated for this case by the new model is shown in Figure 72. The flow field in the region of the nozzle exit is plotted in Figure 73. It is not known why the differences in the two Case 3 solutions resulted, but it is probable that the change in turbulence levels was a direct contributor to the results. This conclusion was based on the results obtained in Case 1, where the new turbulence model generated a cell pressure that was higher than the test data. This equated to a lower level of turbulence in the jet boundary. It was possible that the lower turbulence level raised the cell pressure in this case, causing the diffuser to operate between points A and B in Figure 5 rather than between points B and C.

The wall static pressures down the length of the diffuser compared to data are presented in Figure 74. As in Case 2, the high cell pressure prevents any quantitative conclusions to be drawn.

CASE 4

This case, which used the diffuser having a L/D of 12, provided started diffuser operation by driving the primary nitrogen flow at a total pressure of 460 psia. The cell pressure as a function of iteration count is shown in Figure 75. The final cell pressure was reached in fewer iterations than previously (approximately 25,000 versus 45,000), but the value was higher than the test data by approximately 15 percent.

The flow field generated in this case is presented in Figure 76, and the wall static pressures, both calculated and data, are presented in Figure 77. Unlike the previous two cases, these two graphs shown a definite difference from the unmodified Case 4 run. Whereas the original run provided good agreement with data, the new run showed the same problem that plagued the Case 1 results. Referring between Figure 64 and Figure 77, it was apparent that the new model allowed the shock strength to be overpredicted, resulting in the pressure being recovered much too quickly in the diffuser.

4 SUMMARY AND CONCLUSIONS

The objective of this research was satisfied by the application of the PARC2D program to an axisymmetrical nozzle/diffuser configuration similar to that used in turbine engine and rocket motor testing at the AEDC. The calibration effort used data from available literature and was limited to an axisymmetrical diffuser system that used single specie, single stream, nonreactive flow. The calibration range included a convergent/divergent nozzle with an exit area to throat area ratio of 9, a diffuser with a diffuser area to nozzle throat area ratio of 36 with a diffuser length to diameter ratio of 4, and a diffuser with a diffuser area to nozzle throat area ratio of 16 with a diffuser length to diameter ratio of 12. Calibration parameters selected for the study were diffuser pressure rise ratio and diffuser wall static pressure. The calibration study was broken into three sections. The first part was a parametric study to investigate the characteristics of certain aspects of the PARC2D program and the solution technique. The second section of the research compared PARC2D results to actual test data. A modification was made to the algebraic turbulence model

that permitted an increase in time step size by a factor of five or greater for the cases examined. The effect of this modification on the results was also presented.

The calibration effort demonstrated that the PARC2D program can predict started diffuser pressure rise ratios to within 5 percent of test data for the configurations considered using a value of COFMIX of 0.115. The diffuser configuration having an L/D of 4 and an A_d/A_t ratio of 36 (Case 1) overpredicted the strength of the normal shock in the diffuser, resulting in the diffuser wall static pressures exceeding the test data by as much as 300 percent. For the diffuser system having an L/D of 12 and an A_d/A_t ratio of 16 (Case 4), the PARC2D program predicted the diffuser wall static pressures to within 50 percent of the data. The errors were due mainly to a slight shift in the maximum pressure locations rather than because of incorrect pressure levels. Case 3, the one case considered that operated with the diffuser unstarted and with the nozzle flow underexpanded, could not be converged to a steady cell pressure solution, but was cyclic about the actual test value. Both Case 2 and Case 5 operated with the diffuser unstarted and with the nozzle flow overexpanded. The calculations for both cases overpredicted the cell pressure level by 30 percent or more. Quantitative

comparisons of the predicted diffuser wall static pressures for the three unstarted diffuser cases were not possible due to the improper cell pressure predictions.

The modification to the algebraic turbulence model permitted an increase in the time step size by a factor of 5 or more. Convergence rates were not increased, however. Reanalysis of started diffuser case (Case 1) resulted in a cell pressure prediction 80 percent higher than the test data. Likewise, Case 3 and 4 cell pressure predictions were 60 percent and 15 percent over the test data, respectively. The diffuser wall static pressure predictions for Case 1 and Case 3 did not differ significantly from the previous calculations other than the higher cell pressure levels. Case 4 diffuser wall static pressure predictions indicated that the shock strength in the diffuser was overpredicted, causing a pressure level a maximum of approximately 400 percent higher than the test data.

Four different parameters were investigated in the parametric study. The effect of the cell volume was considered, indicating that it was not necessary to include the entire cell volume in the computational grid. Removing most of the cell volume, however, caused the results that were obtained in the study to vary, indicating that it was indeed possible to reduce the cell volume

too much. The number of grid points in the diffuser was also investigated. No significant benefits were gained by increasing the gridding in the diffuser from 9,000 points to 15,000 points.

In considering the effects of initial conditions on the final solution, two different aspects were studied. First, the effect of the cell initial pressure was considered. It was found that using a cell pressure lower than the final solution provided a faster convergence rate than using an initial pressure higher than the final solution. The second effort looked at the effect of the initial conditions in the diffuser. It was found that there was no difference between using isentropic, one dimensional flow conditions in the diffuser rather than setting the diffuser initial conditions to no-flow values.

The final part of the parametric studies investigated the influence of the turbulence mixing coefficient on the problem solution. Three different values of COFMIX (0.09, 0.11, and 0.13) were used. As expected, the value of COFMIX was significant. As the value was increased, a proportionally lower value of cell pressure was obtained. Since the mixing coefficient controls the level of turbulence in the jet mixing zone (and hence the amount of mass

entrained by the jet), the result was expected and demonstrated that the mixing coefficient could be used to adjust the PARC2D results to match the test data.

The comparison of the PARC2D program results to test data considered both started and unstarted diffuser operation. Two diffuser configurations were included in the study, the first having a L/D of 4 and the second having a L/D of 12. The first case, using the diffuser having a L/D of 4, simulated started diffuser operation and was used to adjust the PARC2D program results to the test data. It was found that a turbulence mixing coefficient value of 0.115 was needed to give the correct cell pressure. Comparing the calculated wall static pressures to test data clearly indicated that the PARC2D program had problems with the flow in the diffuser and overpredicted the shock strength. This resulted in the pressure increasing at a faster rate in the diffuser than the data indicated.

The second and third cases considered also used the diffuser having a L/D of 4 and were used to investigate unstarted diffuser operation. Case 2 operated with a primary fluid total pressure such that the flow separated in the nozzle, and Case 3 provided sufficient total pressure to keep the nozzle flowing full. The results of Case 2

indicated that the PARC2D program overpredicted the cell pressure by over 30 percent. Case 3 results appeared to be converging in on the appropriate cell pressure, but the results were highly cyclic in nature. Because of the large number of iterations required, it was not possible to run the solution to complete convergence. In both cases, the error due to the improper cell pressures prevented any direct comparison between the calculated wall static pressures with the test data.

The fourth case used the diffuser having a L/D of 12 with a primary total pressure sufficiently high enough to start the diffuser. The results effectively fell right on the test data using the value for the turbulence mixing coefficient obtained in Case 1. Unlike Case 1, however, the fourth case also provided agreement with test data to within 50 percent when comparing the wall static pressures down the length of the diffuser. The errors were due mainly to a slight shift in the maximum pressure locations rather than incorrect pressure levels.

The fifth case used the diffuser having a L/D of 12 and provided one additional check of unstated diffuser operation. As in Case 2, the primary total pressure was insufficient to keep the nozzle flowing full. The results were as expected (based on the Case 2 results), with the

cell pressure being 60 percent higher than the test data. It was concluded that the separated flow in the nozzle was the cause of the poor results. The diffuser wall static pressure predictions were not compared due to the incorrect cell pressure predictions.

Because of the large amount of computer time required to obtain the above solutions, a modification to the algebraic turbulence model was made that changed the turbulence mixing coefficient from a constant to a function of the local Mach number. Cases 1, 3, and 4 were reanalyzed to study the effect of this change. In all cases, the maximum time step size, which directly controls the rate of convergence, was increased by a factor of five or more. For Case 1, the PARC2D program predicted a cell pressure 80 percent higher than the test data. The trend of the wall static pressures did not change. Case 3 and 4 likewise indicated some important differences. In Case 3, the flow separated in the nozzle and looked like the results from Case 2. The reason for the change in mode of operation was not known, but could have been caused by the change in turbulence level in the flow. The differences in the Case 4 results occurred with both the cell pressure and the diffuser wall static pressures. The cell pressure was approximately 15 percent higher than the test data.

The diffuser wall static pressures indicated that the PARC2D program was overpredicting the shock strength as in Case 1.

Based on the results obtained from the research, several things were concluded. First, the PARC2D program has demonstrated its ability to predict started diffuser pressure rise ratios with an error approaching 0 percent for systems having a convergent/divergent nozzle with an exit area to throat area ratio of 9 and a diffuser with a diffuser area to nozzle throat area ratio of 36 with a diffuser length to diameter ratio of 4, or a diffuser with a diffuser area to nozzle throat area ratio of 16 with a diffuser length to diameter ratio of 12. Because of the limited extent of this calibration effort in terms of diffuser L/D and A_d/A_t , however, the PARC2D program should not be used blindly for any configuration. It is recommended that the program be calibrated to data obtained from a geometry as close to the system under consideration as possible. Use of the program to determine diffuser wall static pressures for such considerations as structural and thermal design is not recommended until additional work is performed to improve the program results. For unstated diffuser systems with the nozzle flow either overexpanded or underexpanded, the use of the

PARC2D program is also not recommended.

The modification to the algebraic turbulence model allowed the time step size to be increased by a factor of 5 or more for the specific geometries considered. The benefits, however, were not directly apparent since for most cases the same number of iterations were required to obtain a solution. There was also no improvement in the solution accuracy. The model in its present form also requires some additional adjustment of the turbulence equation coefficients to provide a better match to test data.

5 RECOMMENDATIONS

There are several areas where additional work should be conducted to improve the results obtained in this research effort. The most obvious area that requires additional consideration is with the prediction of the pressure recovery mechanism in the diffuser during started operation. The diffuser wall static pressures predicted in Case 1 indicated that the PARC2D program recovered the pressure by a strong normal shock located just downstream of where the nozzle exit flow impinged on the diffuser wall. The test data, however, indicated that the pressure was in fact recovered by a series of weak normal shocks positioned down the length of the diffuser. The results from Case 4 indicated that the PARC2D program correctly mimicked the pressure recovery mechanism in the diffuser. It is not known why the differences between the two cases occurred and a resolution to the problem is needed. Until the PARC2D program can provide accurate and consistent predictions of the conditions along the diffuser wall, the program will provide little additional information over and above that which could be obtained from a much more cost effective and much simpler to apply control volume analysis tool.

Another area that requires additional consideration is the PARC2D program's handling of the flow inside the nozzle during overexpanded operation. The results from Case 2 and Case 5 indicated that the PARC2D program did not allow the flow to separate from the nozzle wall at the appropriate location. This resulted in considerable error in the cell pressure predictions. This is a critical problem, and the use of the PARC2D program for overexpanded nozzle flows is not recommended until this problem is resolved.

Another consideration for additional work includes a look at the diffuser geometries reported in [8] that were not included in this study. The additional knowledge concerning the effect of L/D and A_d/A_t on the PARC2D program performance could help define the program's region of applicability more definitively. Given the differences between the diffuser wall static pressure predictions in Case 1 and Case 4, additional geometries could assist in the resolution of the pressure recovery mechanism problems by more accurately defining where the program works and where it does not work.

As a consideration as to how the convergence rate might be increased, a study of the maximum rate at which the PARC2D program allows certain derivatives to vary from

one time step to the next should be investigated. The work reported herein limited the changes in derivatives to 10 percent. Case runs with the value both lower and higher could provide insight into the effect the rate of change has on the convergence.

It is also recommended that a modification to the PARC2D program be considered that will split the calculation of viscosity between that which is needed to capture shocks and that which is needed in the shear layers. The PARC2D program in its current form does not differentiate between the two. Splitting the calculation could assist in alleviating the errors in wall static pressures that were present in the work reported herein.

Finally, the modification to the algebraic turbulence model which bases the level of turbulence in the flow on the local Mach number requires some additional work. The results presented herein indicate that the model currently underpredicts the diffuser pressure recovery ratio. Some simple tuning of the constants in the model should provide the extra turbulence that is needed to better match test data. The tuning could be accomplished either through direct manipulation of the equation coefficients or by placing a multiplying factor in front of the equation.

BIBLIOGRAPHY

BIBLIOGRAPHY

1. Kroll, A. E. "The Design of Jet Pumps." Chemical Engineering Progress, Vol. 1, No. 2, 1947.
2. Chen, S. T. "Optimization of Jet Entrainment with a Diffuser." Washington State, Ph.D. Dissertation, 1974.
3. Morrison, R. "Jet Ejectors and Augmentation." NACA Advanced Report, 1942.
4. Porter, J. L. and Squyers, R. A. "A Summary/Overview of Ejector Augmentor Theory and Performance, Phase II - Technical Report." ATC Report No. R-91100/9CR-47A, Volume II, Vought Corporation Advanced Technology Center, September 1979.
5. Cambell, S. The Home Water Supply. Storey Communications, Inc., Pownal, Vermont, 1983, pp. 79-80.
6. Taylor, D. "Ejector Design for a Variety of Applications." AEDC Contract No. F40600-69-0001, March 1969.
7. Flügel, G. "The Design of Jet Pumps." NACA TM 982 (Berechnung von Strahlapparaten) VDI-Forschungsheft 395, March - April 1939, pp. 1-21.
8. Fortini, A. "Performance Investigation of a Nonpumping Rocket-Ejector System for Altitude Simulation." NASA TN D-257, October 1959.
9. German, R. C. and Bauer, R. C. "Effects of Diffuser Length on the Performance of Ejectors without Induced Flow." AEDC-TN-61-89 (AD262888), August 1961.
10. Hill, J. A. F., Baron, J. R., Schindell, L. H., and Harkham, J. R. "Mach Number Measurements in High Speed Wind Tunnels." AGARDOGRAPH 22, October 1952.
11. Dean, Robert C. Jr. "Aerodynamic Measurements." Gas Turbine Laboratory, Massachusetts Institute of Technology, 1953.
12. Weeks, R. M., Terry, J. E., and Horn, D. D. "Calibration of a 26 by 26 in. Semi-Flexible Variable Mach Number Nozzle in the Engine Test Facility Test Cell T-4." AEDC-TM-59-2, January 1959.

13. Latvala, E. K., Anderson, T. P., and Clark, H. K. "Calibration of a 6 by 6 inch Semi-Flexible Variable Mach Number Nozzle in the Engine Test Facility Test Cell T5A-1." AEDC-TM-58-24, December 1958.
14. Matz, R. J., Neely, James, Nygaard, R. C., and Ward, T. R. "J2 Nozzle Model Aerodynamic Documentation Test." AEDC-TM-57-43, July 1957.
15. Phares, W. J., Cooper, G. K., Swafford, T. W., and Jones, R. R. "Application of Computational Fluid Dynamics to Test Facility and Experiment Design." AIAA-86-1733, June 1986.
16. Cooper, G. K., Garrard, G. D., and Phares, W. J. "PARC Code Validation for Propulsion Flows." AEDC-TR-88-32, January 1989.
17. Bradley, R. G. "CFD Validation Philosophy." Paper No. 1., AGARD Symposium on Validation of Computational Fluid Dynamics, May 1988, Lisbon, Portugal.
18. Marvin, J. G. "Accuracy Requirements and Benchmark Experiments for CFD Validation." NASA TM 100087, May, 1988.
19. Povinelli, L. A. "CFD Validation Experiments for Internal Flows." NASA TM 100797, May 1988.
20. Bobbitt, P. J. "The Pros and Cons of Code Validation." NASA TM 100657, July 1988.
21. Kutler, P. "A Perspective of Computational Fluid Dynamics." NASA TM 88246, May, 1986.
22. Huddleston, D. H., Cooper, G. K., and Phares, W. J. "A Computational Fluid Dynamics Evaluation of Test Cell Recirculation Effects on High-Bypass Turbofan Engine Surface Pressure Distributions." AIAA-86-1384, AIAA/ASME/SAE/ASEE 22nd Propulsion Conference, Huntsville, Alabama, June 16-18, 1986.
23. Peters, C. E. and Phares, W. J. "Integral Analysis of Ducted Two-Stream Mixing with Recirculation." AEDC-TR-77-115, March 1978.

24. Hunter, L. G. and Kent, T. J. "Inlet Analysis Using Computational Fluid Dynamics." AIAA-86-2611, AIAA/AHS/ASEE Aircraft Systems, Design and Technology Meeting, Dayton, Ohio, October 20-22, 1986.
25. Semmes, R. G., Arbiter, D. G., and Dyer, R. D. "Efforts Toward the Validation of a Computational Fluid Dynamics Code for Analysis of Internal Aerodynamics." AGARD N89-18643, December 1988.
26. Soni, B. K. "Two- and Three-Dimensional Grid Generation for Internal Flow Applications of Computational Fluid Dynamics." AIAA Paper 85-1526, AIAA 7th Computational Fluid Dynamics Conference, Cincinnati, Ohio, 1985.
27. Shapiro, A. H. The Dynamics and Thermodynamics of Compressible Fluid Flow. The Ronald Press Company, New York, New York, 1953, pg. 135.
28. Thomas, P. D. "Numerical Method for Predicting Flow Characteristics and Performance of Nonaxisymmetric Nozzles - Theory." Langley Research Center, NASA CR 3147, September 1979.
29. Baldwin, B. S. 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 1978.
30. Cooper, G. K. and Sirbaugh, J. R. "The PARC Distinction: A Practical Flow Simulator." AIAA Paper No. 90-2002, To be presented at the 26th Joint Propulsion Conference, Orlando, Florida, July 1990.
31. 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. 87-100.
32. 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.

33. Pulliam, T. H. "Euler and Thin Layer Navier-Stokes Codes: ARC2D, ARC3D." Notes for Computational Fluid Dynamics User's Workshop, The University of Tennessee Space Institute, Tullahoma, Tennessee, (UTSI Publication E02-4005-023-84), March 12-16, 1984, pp. 15.1-15.85.

APPENDIXES

APPENDIX A

FIGURES

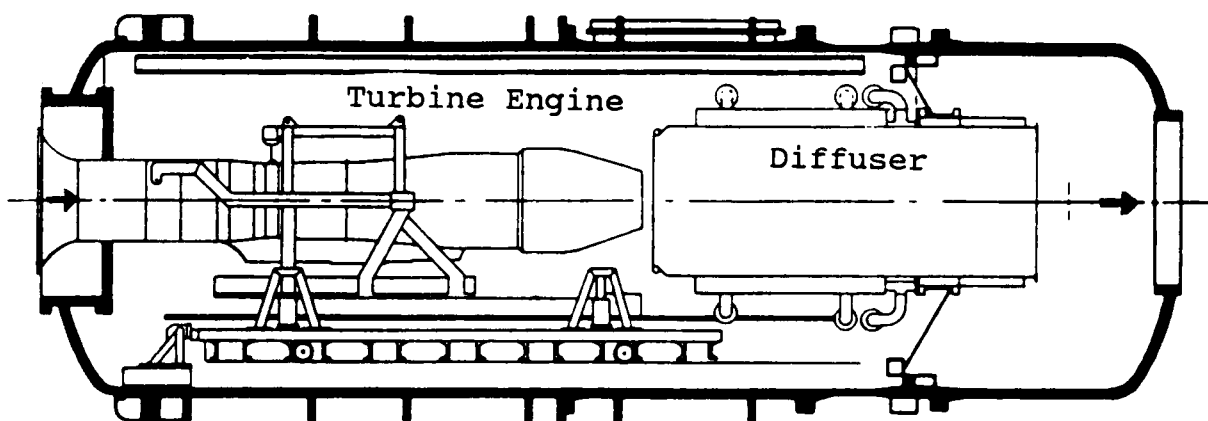


Figure 1. Typical turbine engine test installation

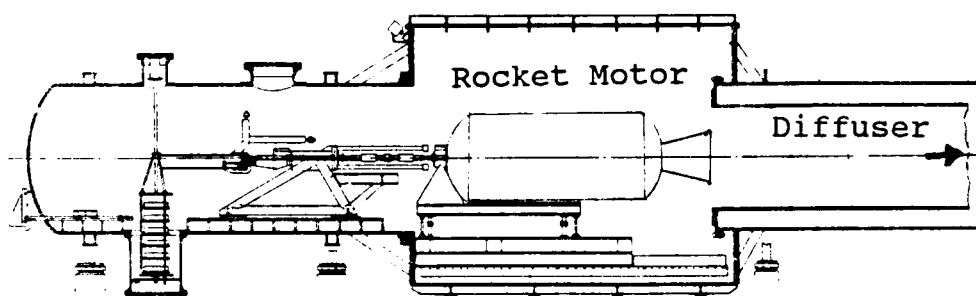


Figure 2. Typical rocket motor test installation

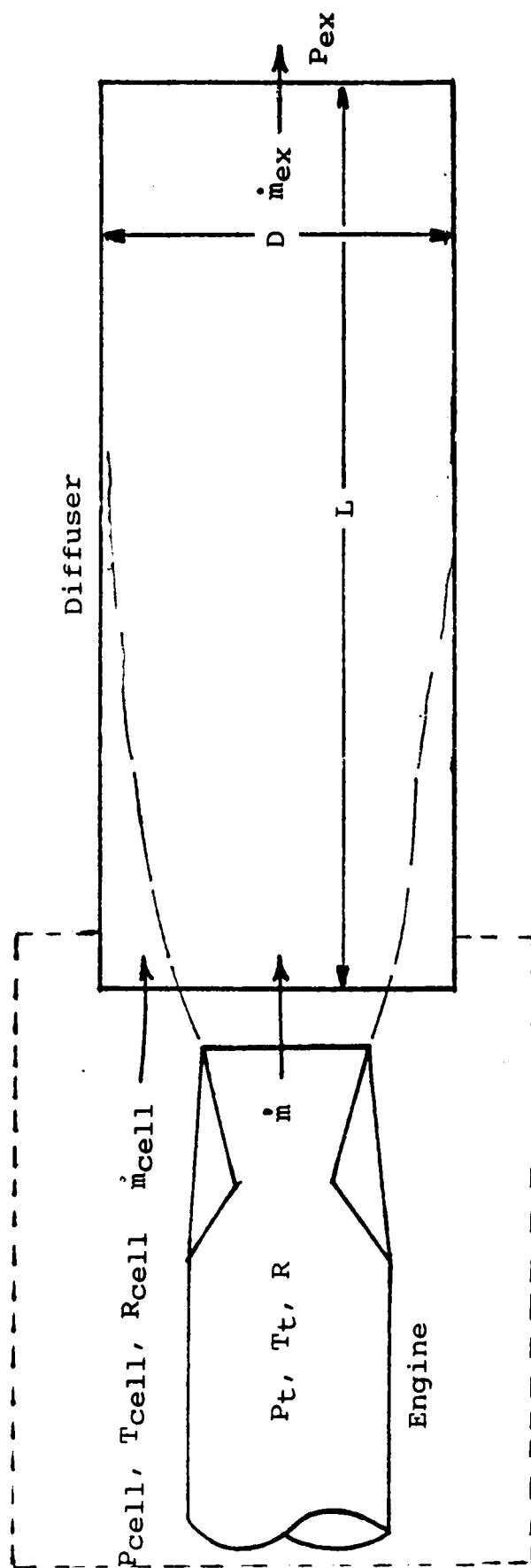
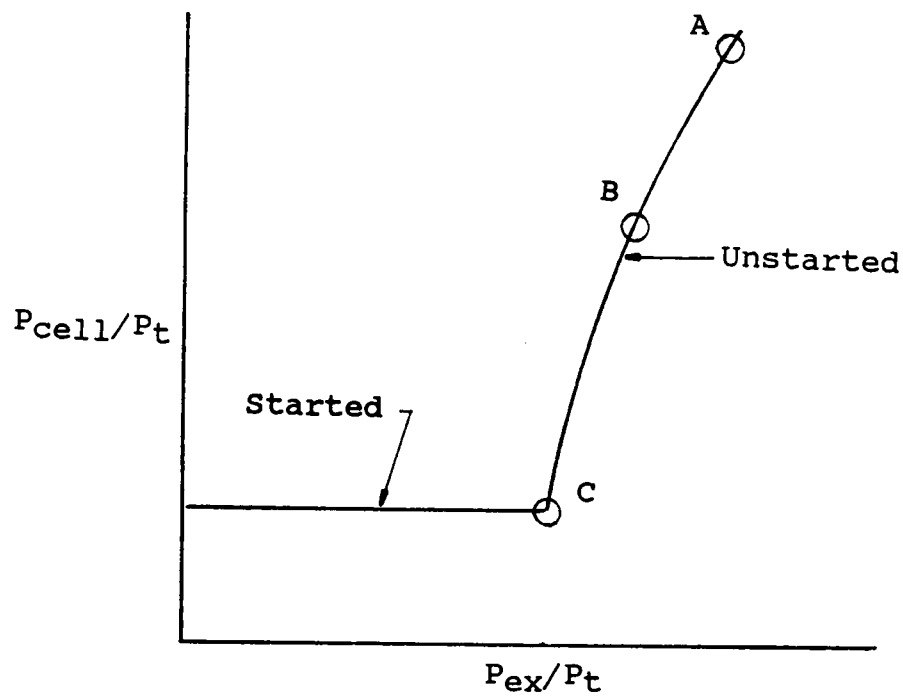
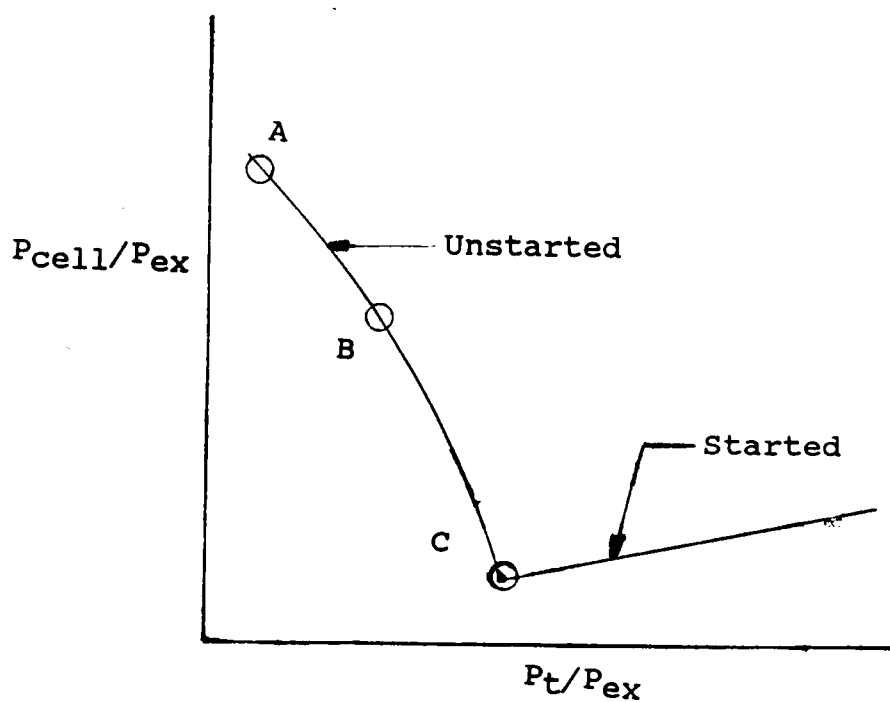


Figure 3. Parameters of interest for diffuser systems



a. P_{cell}/P_t versus P_{ex}/P_t



b. P_{cell}/P_{ex} versus P_t/P_{ex}

Figure 4. Graphical presentation of diffuser performance

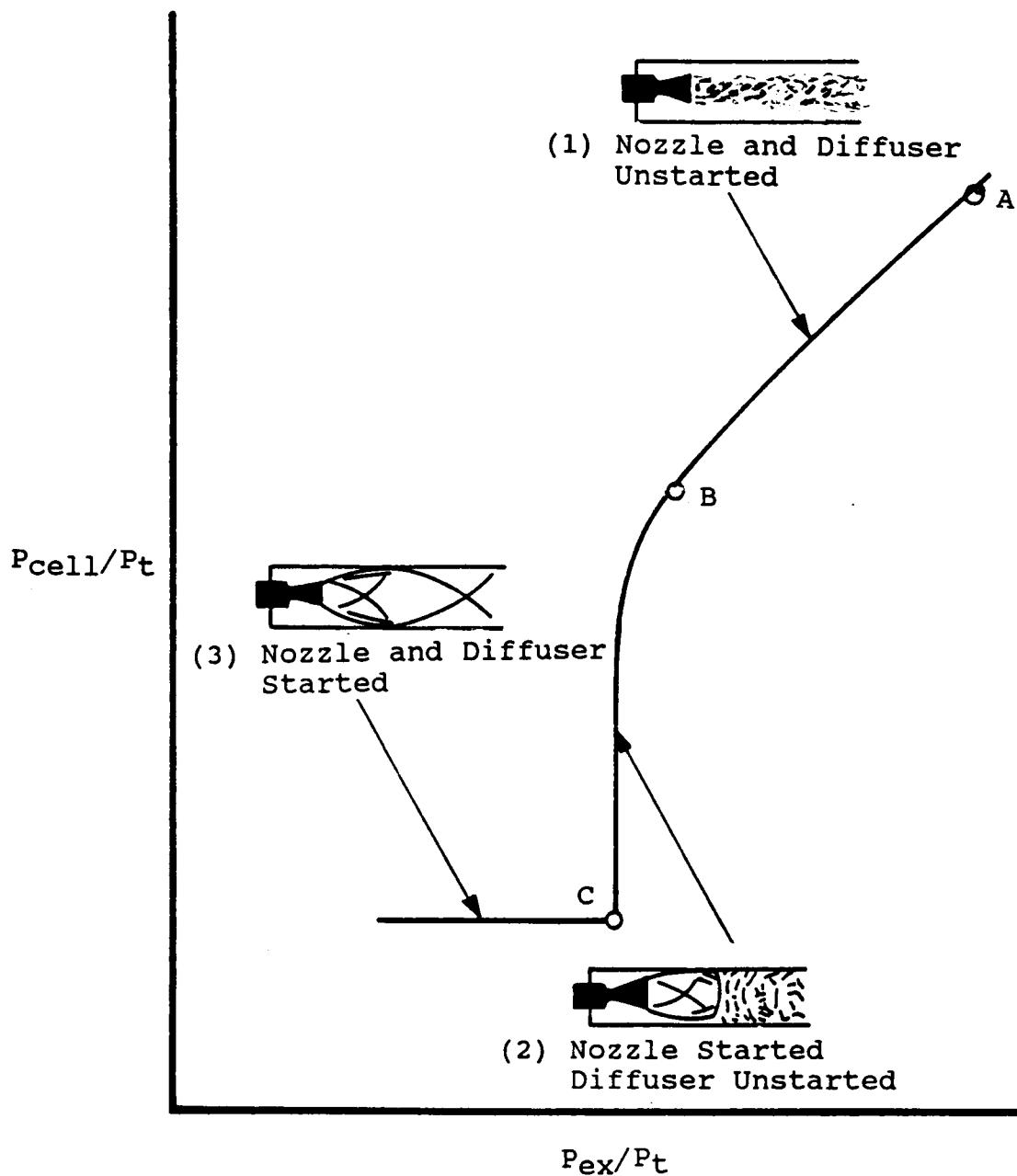


Figure 5. Flow in the diffuser for different operating conditions

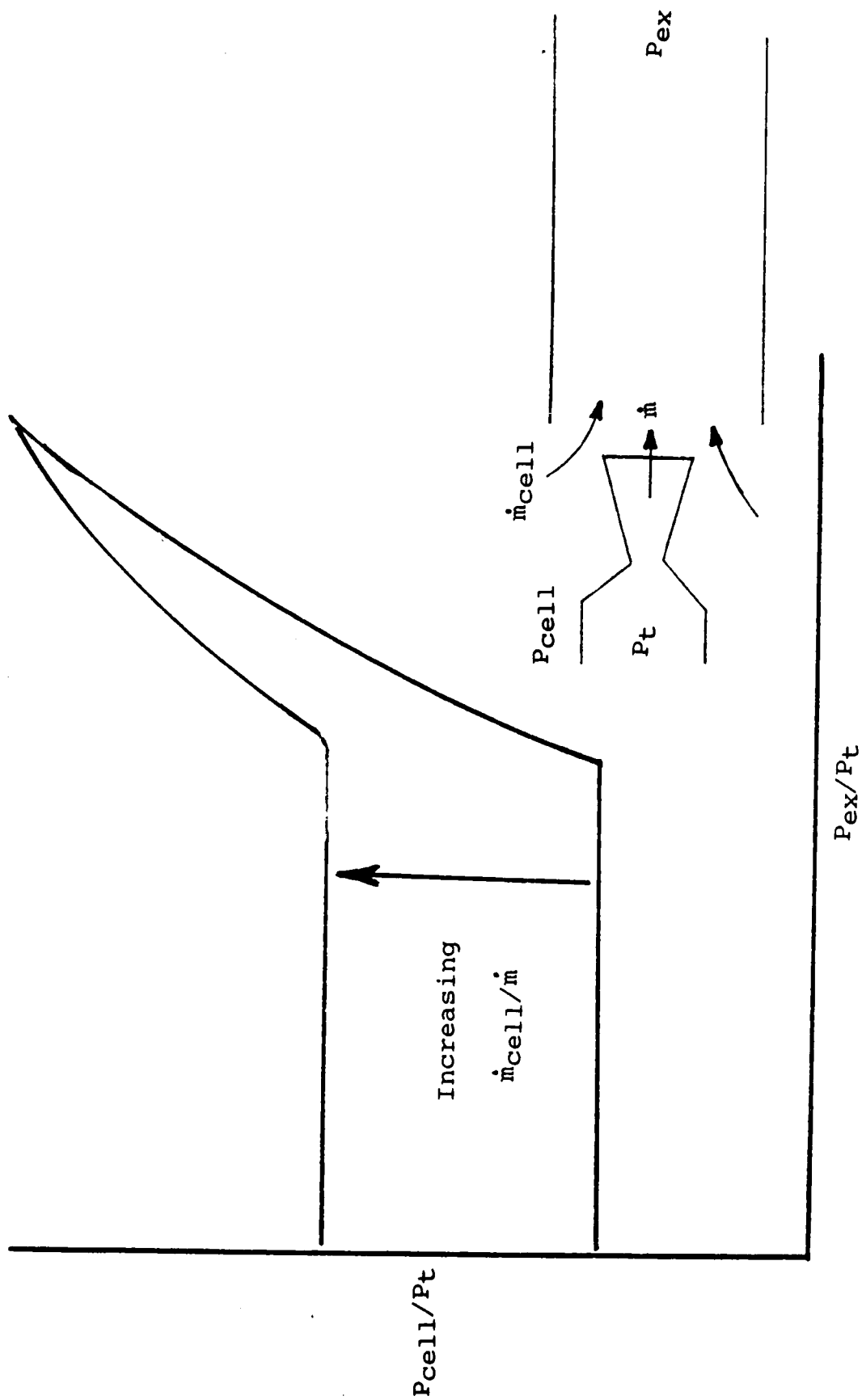


Figure 6. Effect of secondary flow on diffuser performance

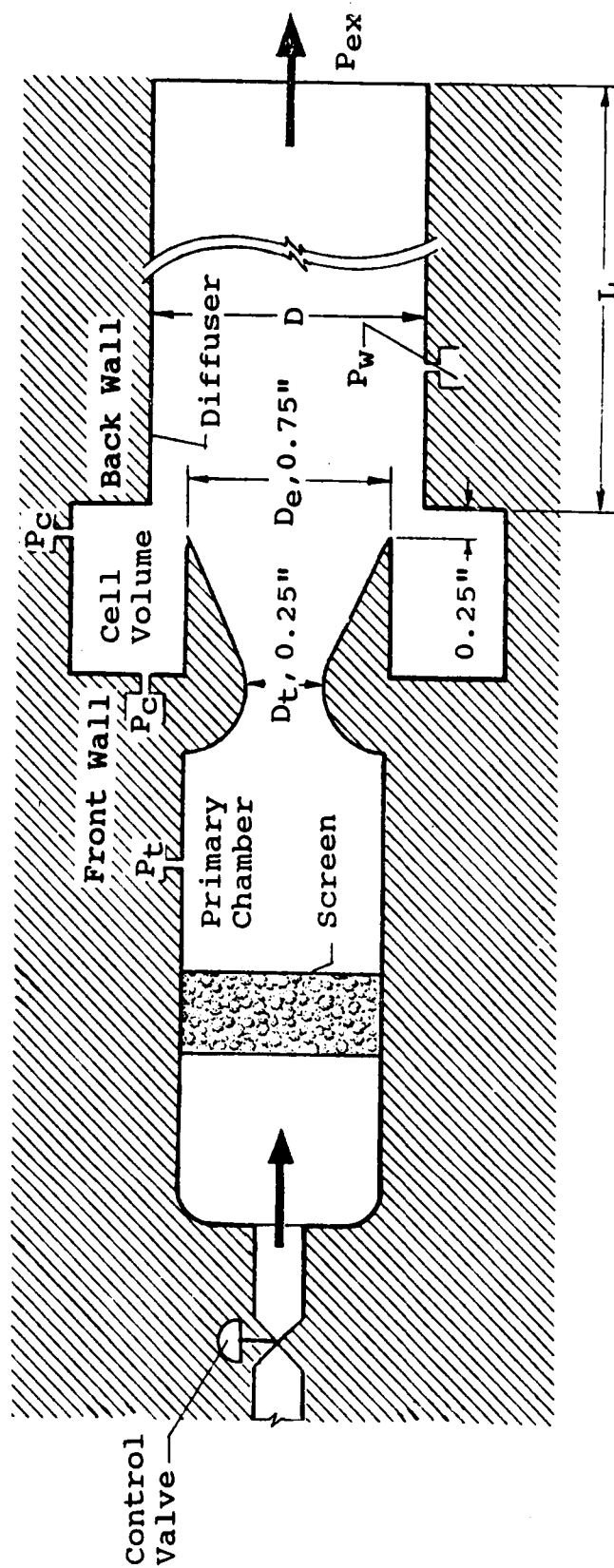


Figure 7. Test apparatus used to obtain calibration data

Source: Fortini, A. "Performance Investigation of a Nonpumping Rocket-Ejector System for Altitude Simulation." NASA TN D-257, October 1959.

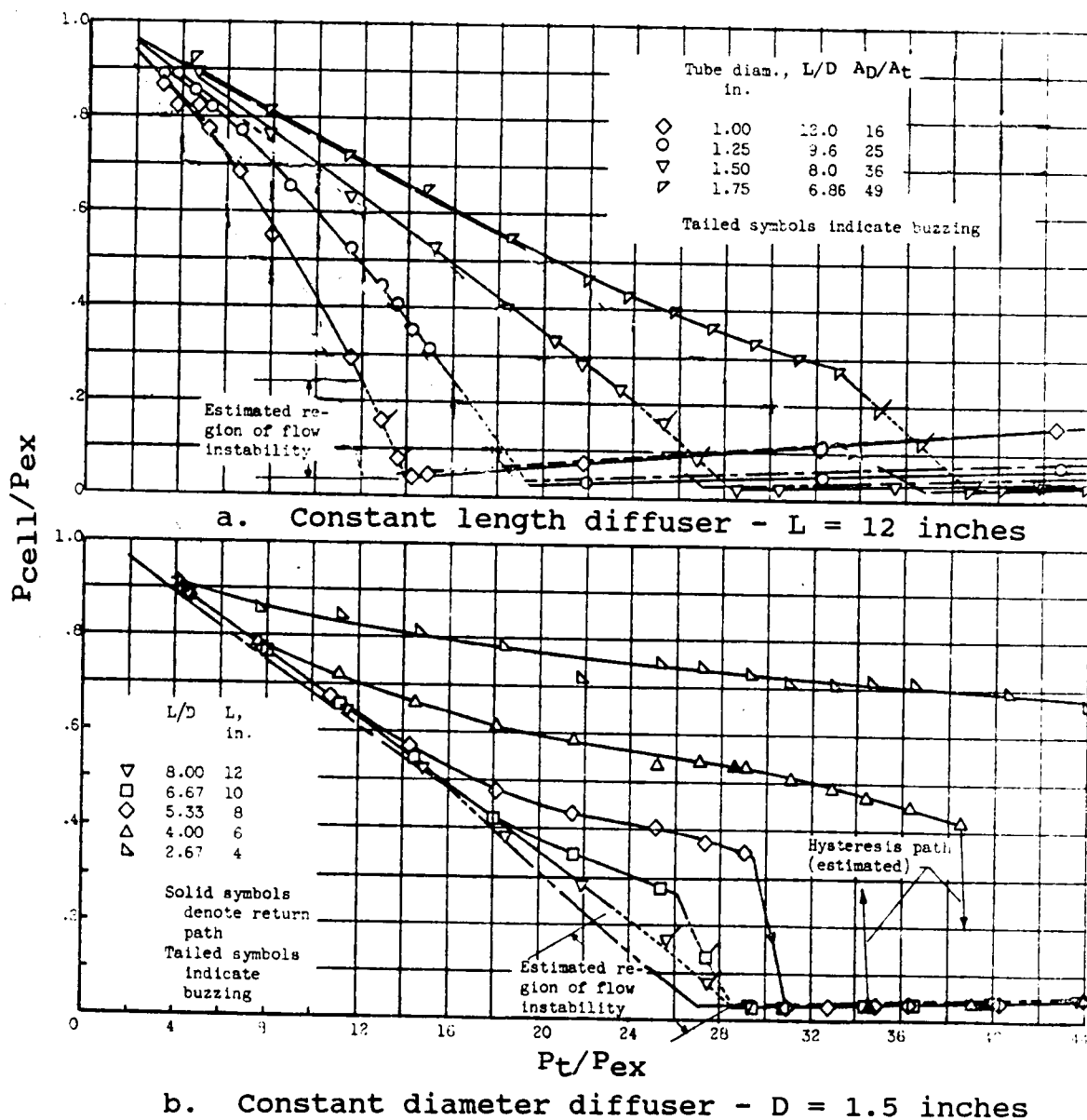


Figure 8. Experimental performance of cylindrical diffuser for various diffuser to nozzle throat area ratios (A_D/A_t) and diffuser length to diameter ratios (L/D)

Adopted From: Fortini, A. "Performance Investigation of a Nonpumping Rocket-Ejector System for Altitude Simulation." NASA TN D-257, October 1959.

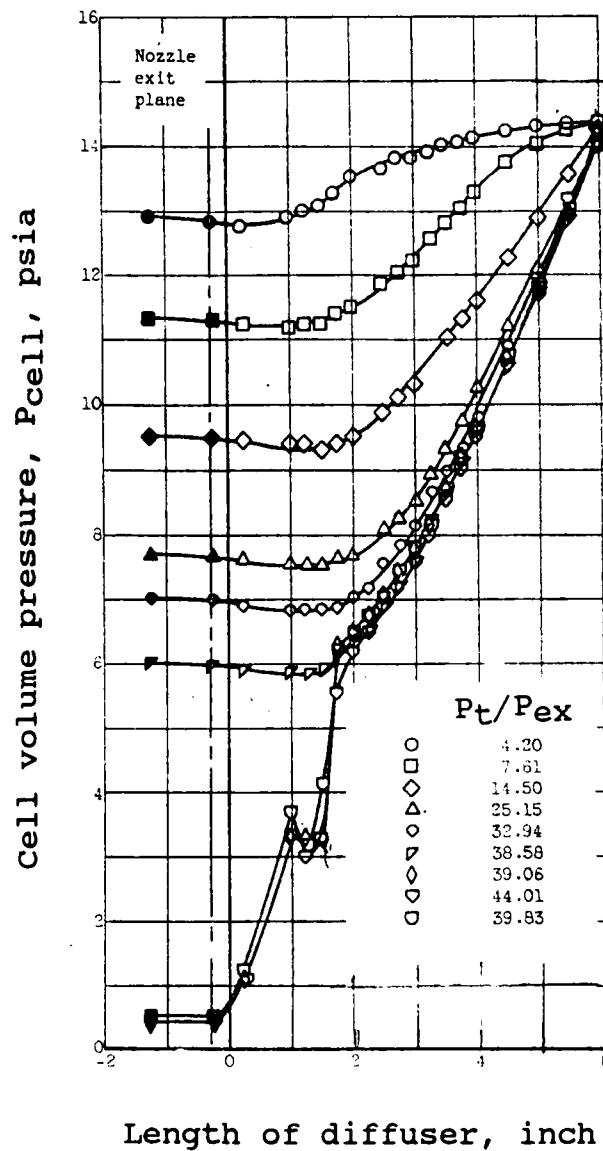


Figure 9. Wall static pressure for 1.5 inch diameter, 6 inch length diffuser at various total pressures (P_t)

Source: Fortini, A. "Performance Investigation of a Nonpumping Rocket-Ejector System for Altitude Simulation." NASA TN D-257, October 1959.

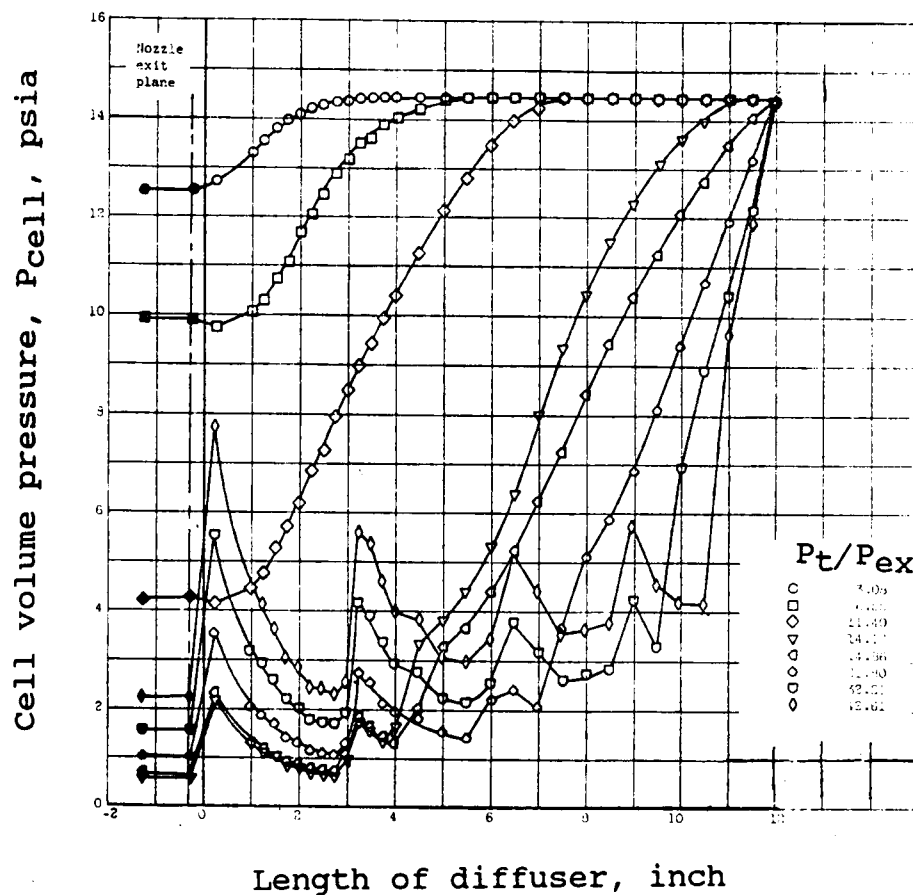


Figure 10. Wall static pressures for 1.0 inch diameter, 12 inch length diffuser at various total pressures (P_t)

Source: Fortini, A. "Performance Investigation of a Nonpumping Rocket-Ejector System for Altitude Simulation." NASA TN D-257, October 1959.

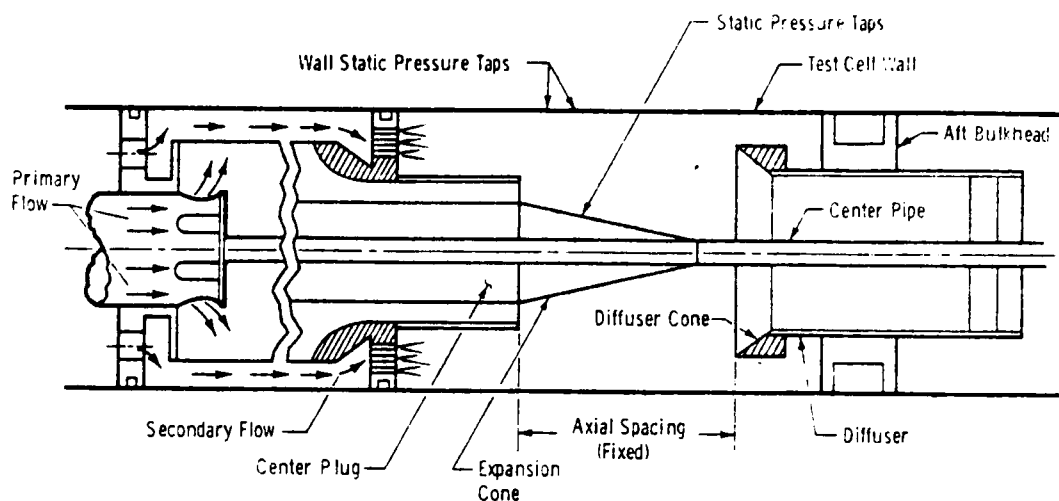


Figure 11. Subscale experimental configuration

Source: Huddleston, D. H., Cooper, G. K., and Phares, W. J. "A Computational Fluid Dynamics Evaluation of Test Cell Recirculation Effects on High-Bypass Turbofan Engine Surface Pressure Distributions." AIAA-86-1384, AIAA/ASME/SAE/ASEE 22nd Propulsion Conference, Huntsville, Alabama, June 16-18, 1986.

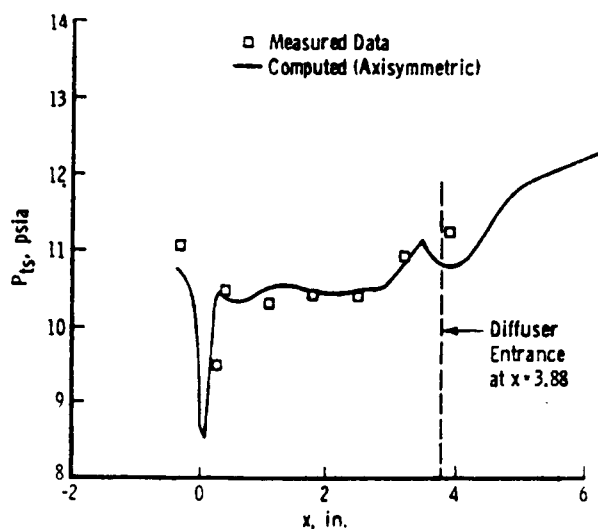


Figure 12. Tube surface static pressure (P_{ts}) versus distance from nozzle exit plane (x) for subscale test

Source: Huddleston, D. H., Cooper, G. K., and Phares, W. J. "A Computational Fluid Dynamics Evaluation of Test Cell Recirculation Effects on High-Bypass Turbofan Engine Surface Pressure Distributions." AIAA-86-1384, AIAA/ASME/SAE/ASEE 22nd Propulsion Conference, Huntsville, Alabama, June 16-18, 1986.

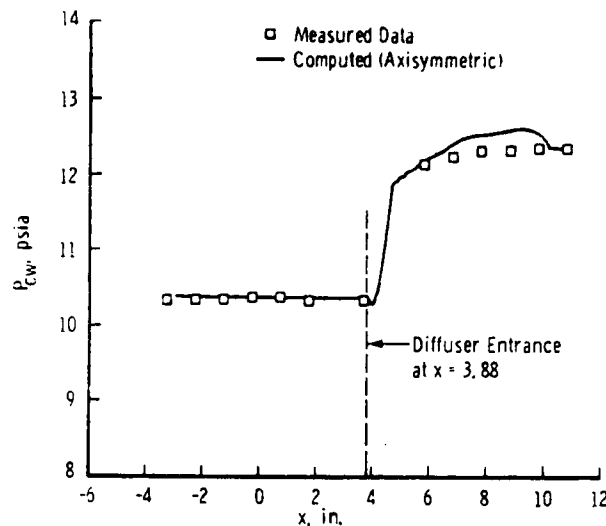


Figure 13. Cell wall static pressure (P_{cw}) versus distance from nozzle exit plane (x) for subscale test

Source: Huddleston, D. H., Cooper, G. K., and Phares, W. J. "A Computational Fluid Dynamics Evaluation of Test Cell Recirculation Effects on High-Bypass Turbofan Engine Surface Pressure Distributions." AIAA-86-1384, AIAA/ASME/SAE/ASEE 22nd Propulsion Conference, Huntsville, Alabama, June 16-18, 1986.

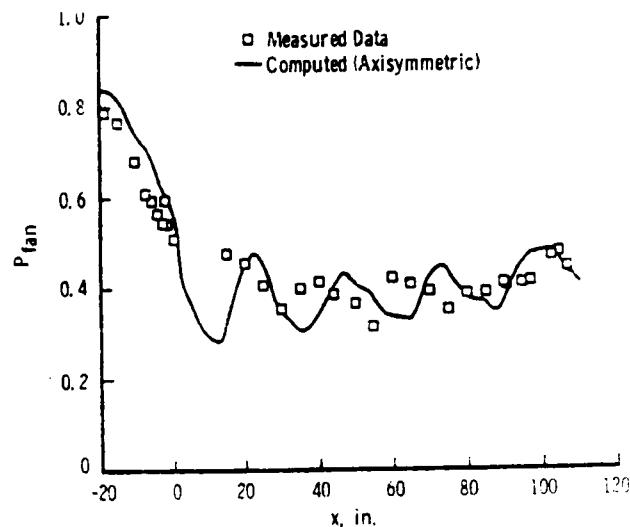


Figure 14. Fan cowl surface pressure (P_{fan}) versus distance from fan stream exit plane (x) for engine/test cell

Source: Huddleston, D. H., Cooper, G. K., and Phares, W. J. "A Computational Fluid Dynamics Evaluation of Test Cell Recirculation Effects on High-Bypass Turbofan Engine Surface Pressure Distributions." AIAA-86-1384, AIAA/ASME/SAE/ASEE 22nd Propulsion Conference, Huntsville, Alabama, June 16-18, 1986.

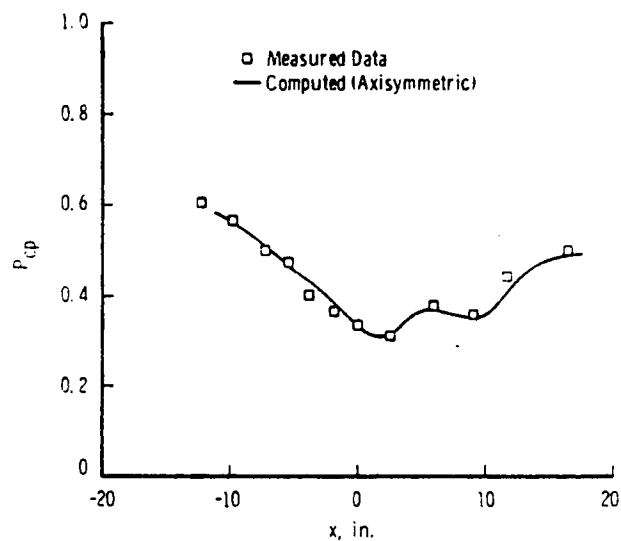


Figure 15. Core plug surface pressure (P_{cp}) versus distance from primary jet exit plane (x) for engine/test cell

Source: Huddleston, D. H., Cooper, G. K., and Phares, W. J. "A Computational Fluid Dynamics Evaluation of Test Cell Recirculation Effects on High-Bypass Turbofan Engine Surface Pressure Distributions." AIAA-86-1384, AIAA/ASME/SAE/ASEE 22nd Propulsion Conference, Huntsville, Alabama, June 16-18, 1986.

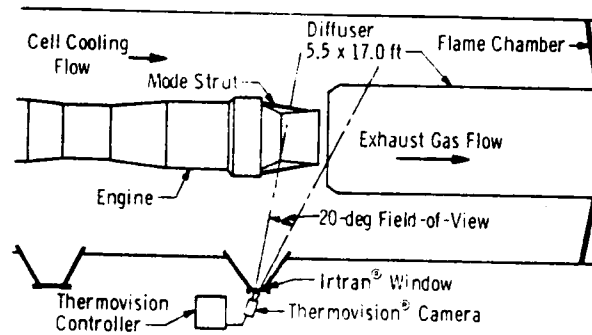


Figure 16. Test configuration for the study of cell heating

Source: Phares, W. J., Cooper, G. K., Swafford, T. W., and Jones, R. R. "Application of Computational Fluid Dynamics to Test Facility and Experiment Design." AIAA-86-1733, June 1986.

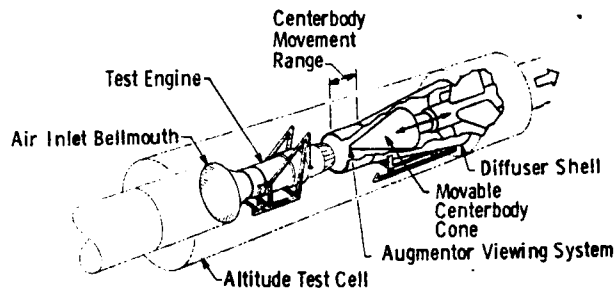


Figure 17. Variable Area Ejector test configuration

Source: Phares, W. J., Cooper, G. K., Swafford, T. W., and Jones, R. R. "Application of Computational Fluid Dynamics to Test Facility and Experiment Design." AIAA-86-1733, June 1986.

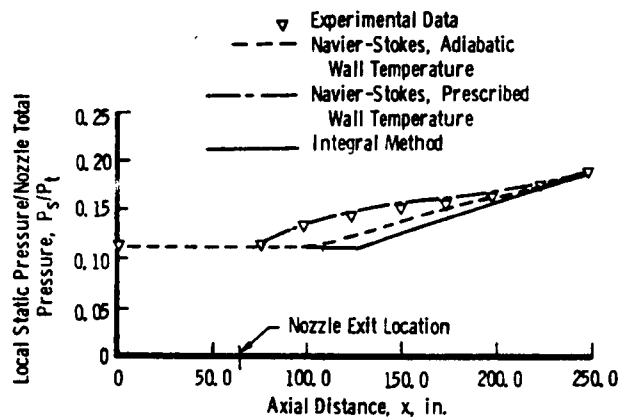


Figure 18. Cell and wall pressures compared to experimental data and integral method results

Source: Phares, W. J., Cooper, G. K., Swafford, T. W., and Jones, R. R. "Application of Computational Fluid Dynamics to Test Facility and Experiment Design." AIAA-86-1733, June 1986.

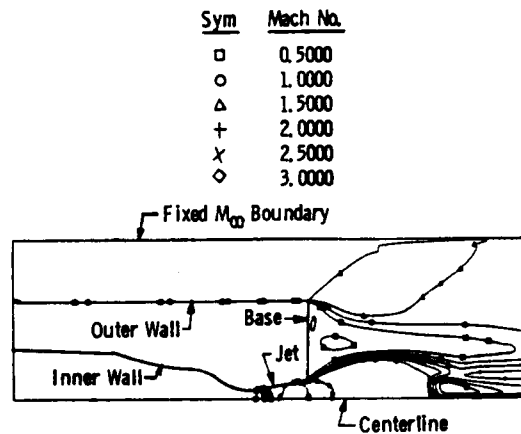


Figure 19. Base flow test case geometry and computed Mach number contours

Source: Phares, W. J., Cooper, G. K., Swafford, T. W., and Jones, R. R. "Application of Computational Fluid Dynamics to Test Facility and Experiment Design." AIAA-86-1733, June 1986.

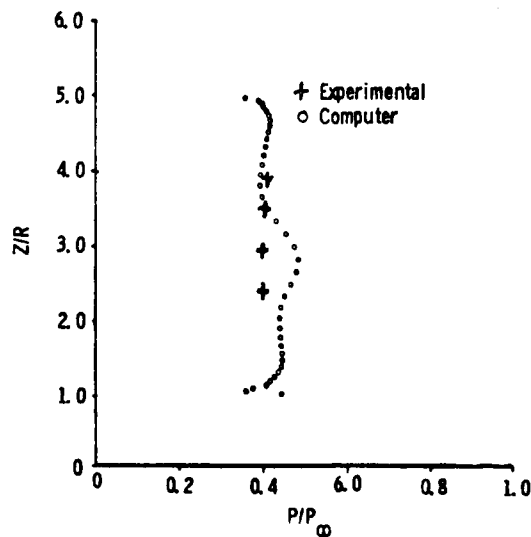


Figure 20. Base flow test case computed versus experimental base static to total pressure ratio (P/P_0)

Source: Phares, W. J., Cooper, G. K., Swafford, T. W., and Jones, R. R. "Application of Computational Fluid Dynamics to Test Facility and Experiment Design." AIAA-86-1733, June 1986.

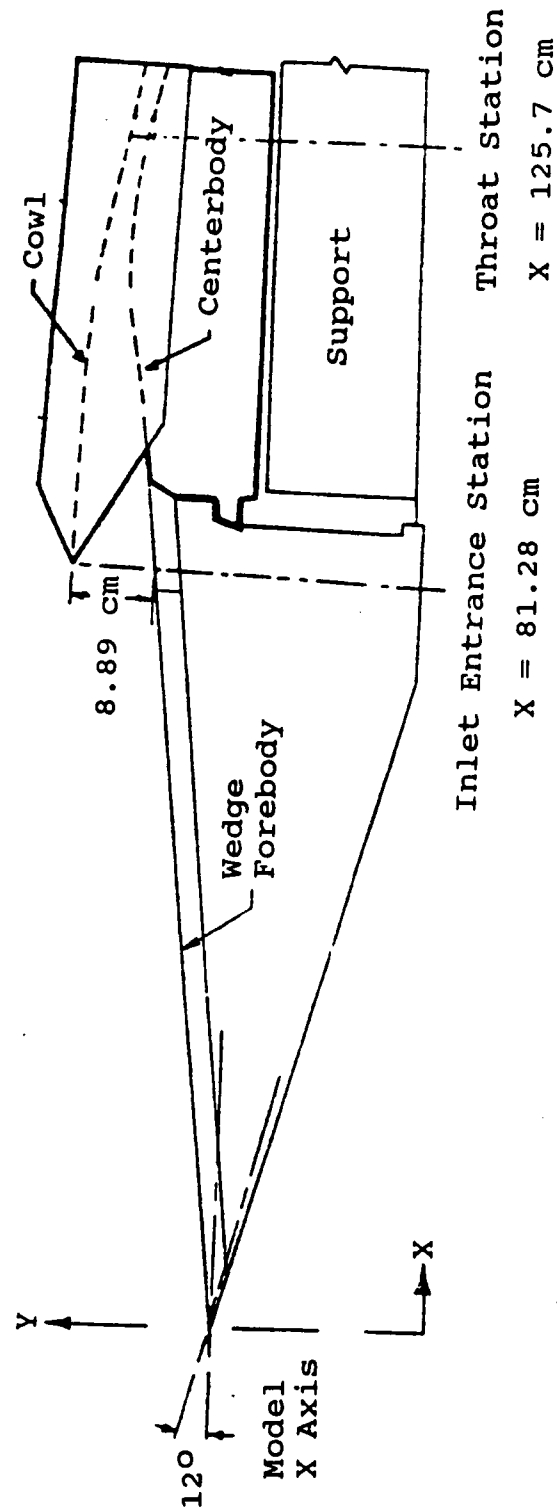
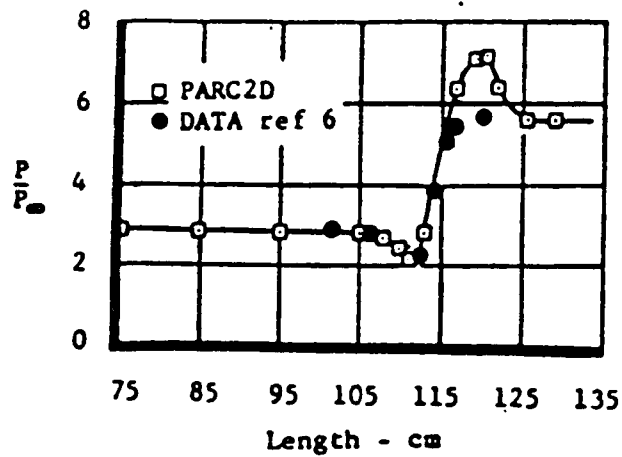
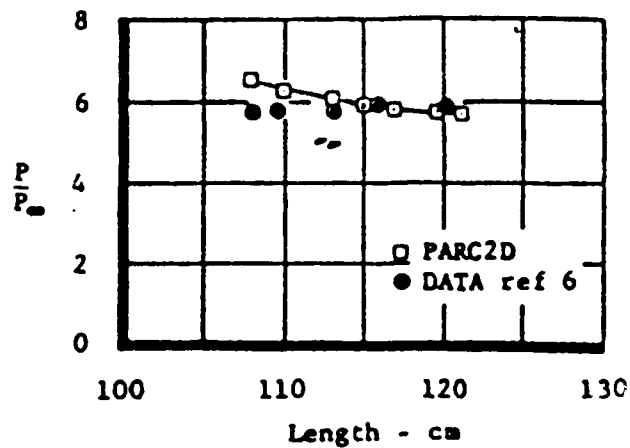


Figure 21. Hypersonic inlet test hardware (P-2 inlet is shown. P-12 is slightly different)

Source: Hunter, L. G. and Kent, T. J. "Inlet Analysis Using Computational Fluid Dynamics." AIAA-86-2611, AIAA/AHS/ASEE Aircraft Systems, Design and Technology Meeting, Dayton, Ohio, October 20-22, 1986.



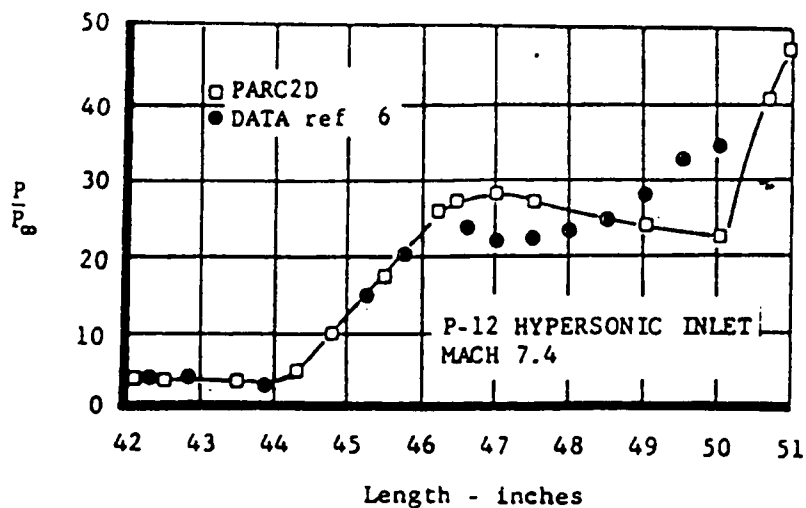
a. Centerbody static pressure for P-2 inlet



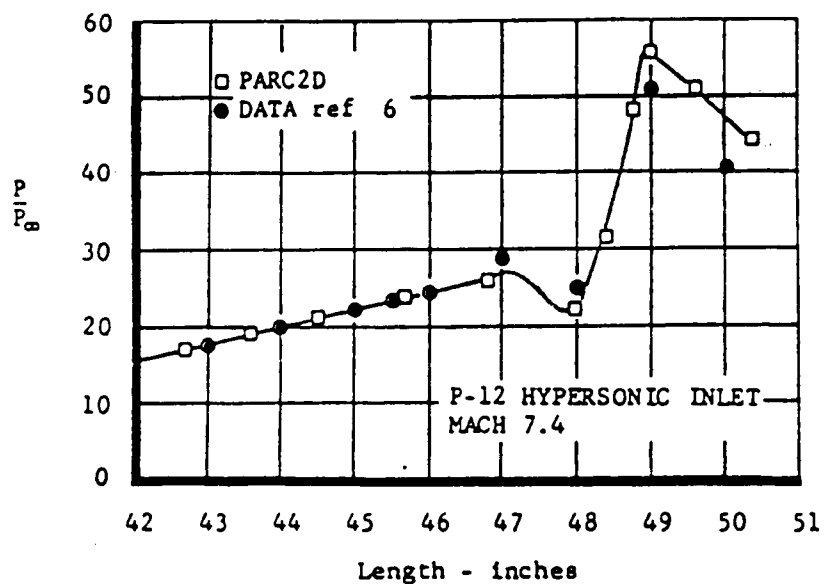
b. Cowl static pressure for P-2 inlet

Figure 22. Comparison of PARC2D results to data at Mach 7.4

Source: Hunter, L. G. and Kent, T. J. "Inlet Analysis Using Computational Fluid Dynamics." AIAA-86-2611, AIAA/AHS/ASEE Aircraft Systems, Design and Technology Meeting, Dayton, Ohio, October 20-22, 1986.



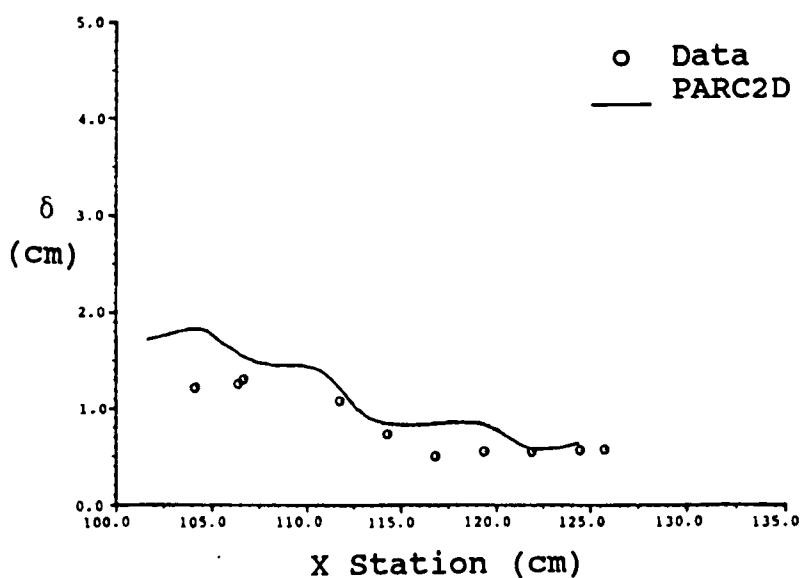
c. Centerbody static pressure for P-12 inlet



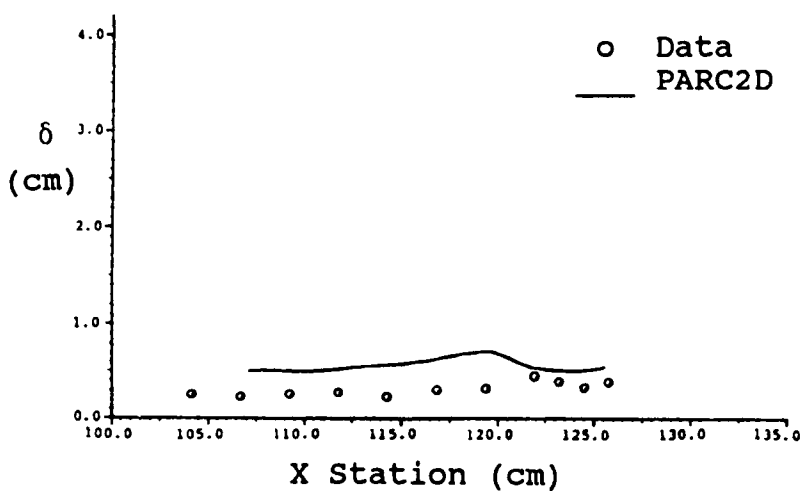
d. Cowl static pressure for P-12 inlet

Figure 22. Comparison of PARC2D results to data at Mach 7.4 (concluded)

Source: Hunter, L. G. and Kent, T. J. "Inlet Analysis Using Computational Fluid Dynamics." AIAA-86-2611, AIAA/AHS/ASEE Aircraft Systems, Design and Technology Meeting, Dayton, Ohio, October 20-22, 1986.



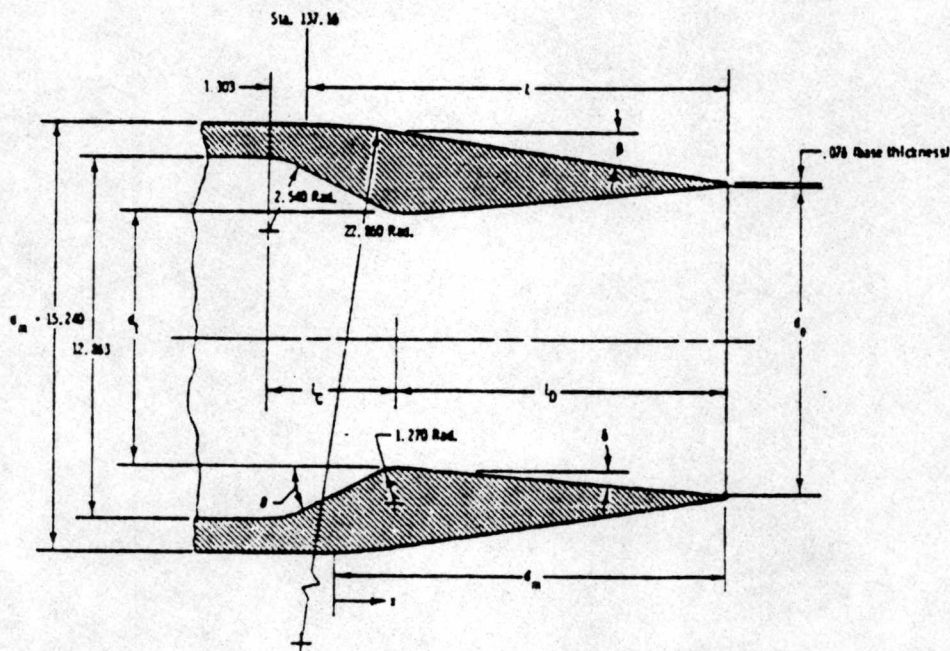
a. Centerbody boundary layer thickness



b. Cowl boundary layer thickness

Figure 23. Boundary layer growth in a hypersonic inlet

Source: Semmes, R. G., Arbiter, D. G., and Dyer, R. D. "Efforts Toward the Validation of a Computational Fluid Dynamics Code for Analysis of Internal Aerodynamics." AGARD N89-18643, December 1988.



$(P_t/P_\infty)_{des}$	A_e/A_t	d_t/d_m	l_c/d_m	l_d/d_m	θ, deg	δ, deg	l/d_m	d_e/d_m
4.25	1.25	0.5	0.286	0.8	42.35	2.12	1.0	0.559

Figure 24. Geometry for the convergent-divergent nozzle

Source: Semmes, R. G., Arbiter, D. G., and Dyer, R. D. "Efforts Toward the Validation of a Computational Fluid Dynamics Code for Analysis of Internal Aerodynamics." AGARD N89-18643, December 1988.

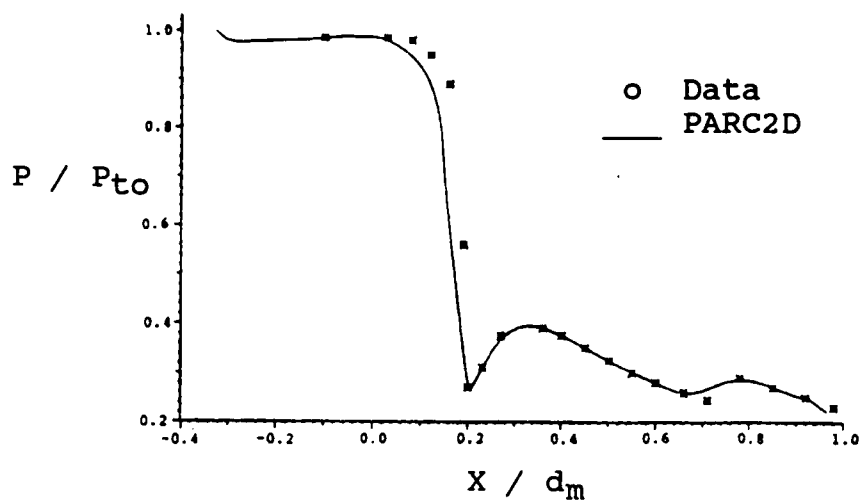


Figure 25. Surface static pressure for the convergent-divergent nozzle

Source: Semmes, R. G., Arbiter, D. G., and Dyer, R. D. "Efforts Toward the Validation of a Computational Fluid Dynamics Code for Analysis of Internal Aerodynamics." AGARD N89-18643, December 1988.

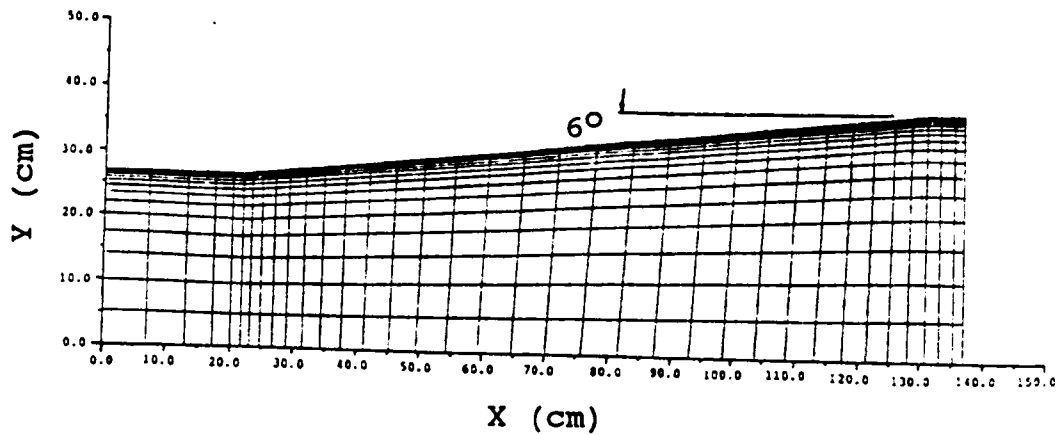


Figure 26. Subsonic diffuser geometry

Source: Semmes, R. G., Arbiter, D. G., and Dyer, R. D. "Efforts Toward the Validation of a Computational Fluid Dynamics Code for Analysis of Internal Aerodynamics." AGARD N89-18643, December 1988.

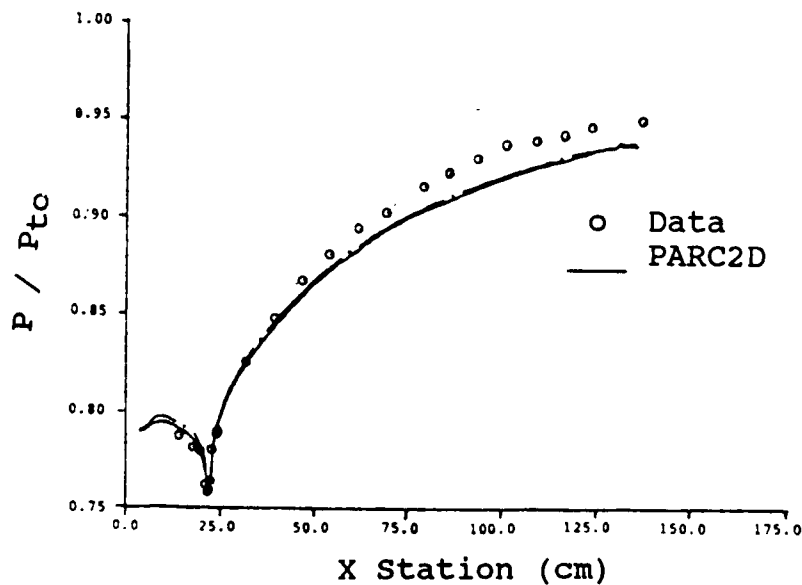


Figure 27. Surface static pressure for the subsonic diffuser

Source: Semmes, R. G., Arbiter, D. G., and Dyer, R. D. "Efforts Toward the Validation of a Computational Fluid Dynamics Code for Analysis of Internal Aerodynamics." AGARD N89-18643, December 1988.

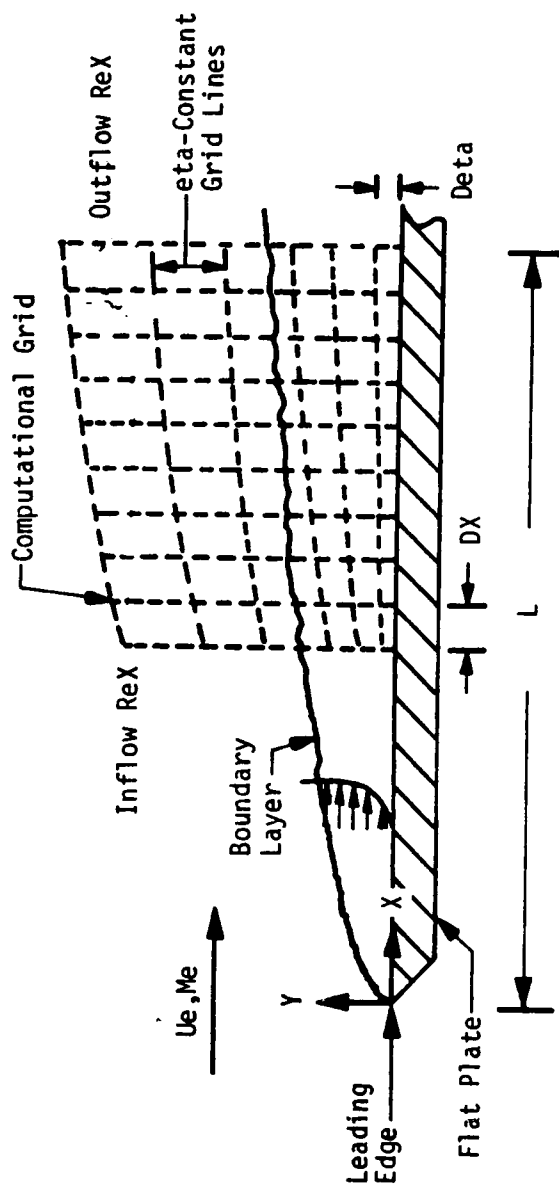


Figure 28. Geometry and basic formulation for flat plate calibration case

Source: Cooper, G. K., Garrard, G. D., and Phares, W. J. "PARC Code Validation for Propulsion Flows." AEDC-TR-88-32, January 1989.

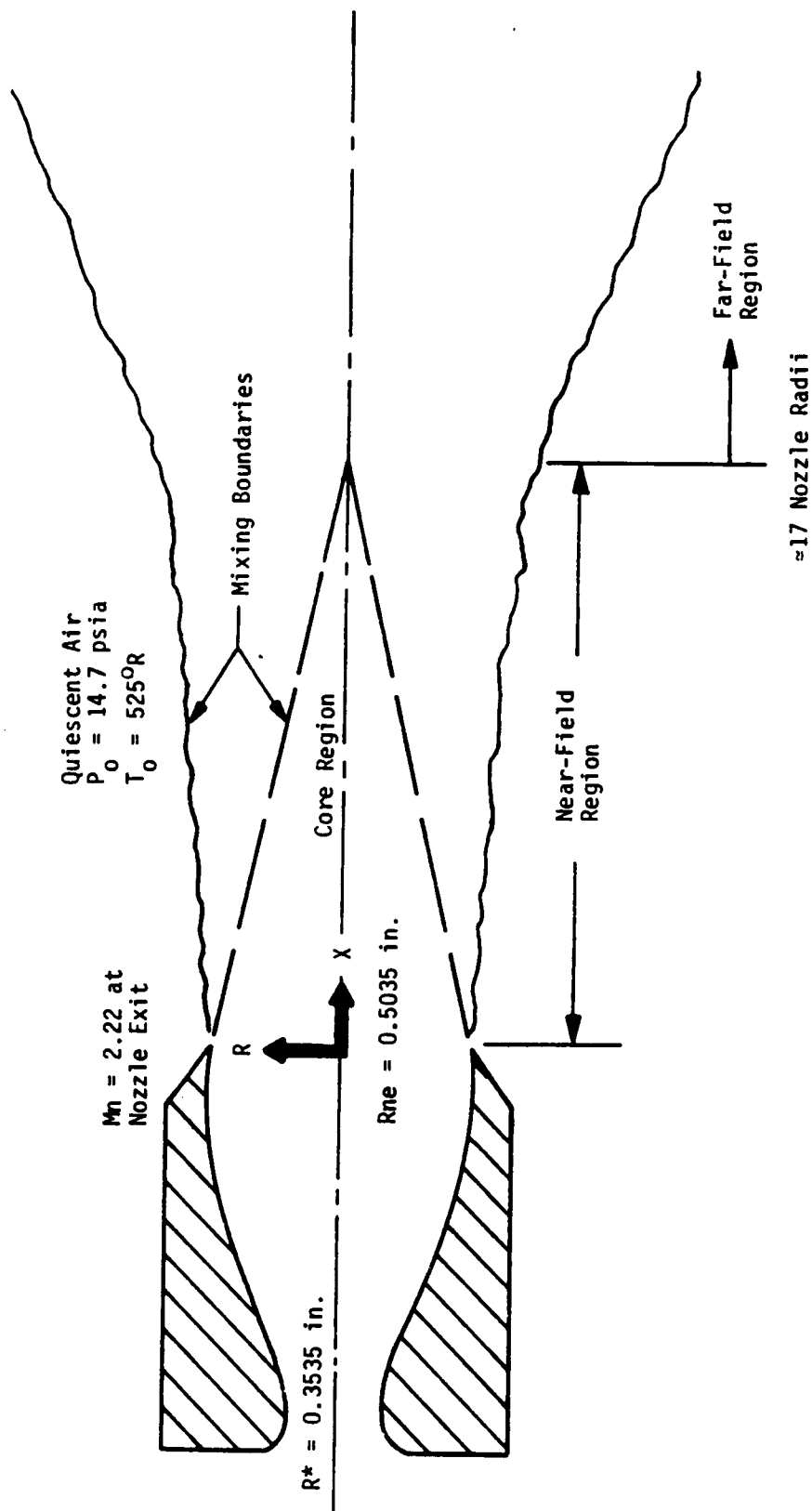


Figure 29. Supersonic nozzle flow basic formulation

Source: Cooper, G. K., Garrard, G. D., and Phares, W. J. "PARC Code Validation for Propulsion Flows." AEDC-TR-88-32, January 1989.

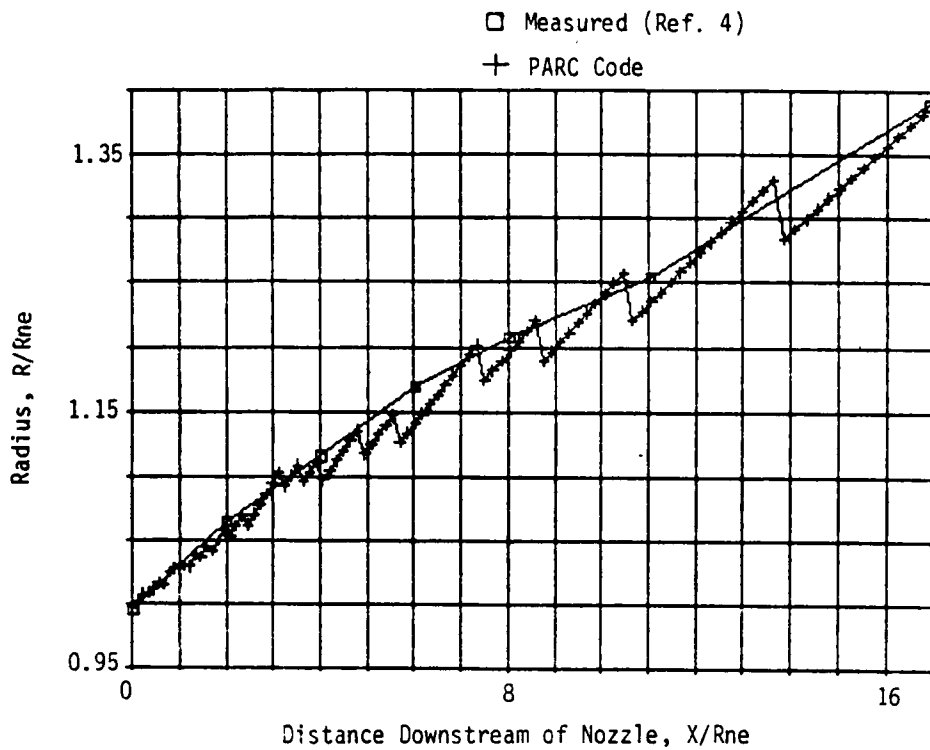


Figure 30. Near field results for supersonic nozzle

Source: Cooper, G. K., Garrard, G. D., and Phares, W. J. "PARC Code Validation for Propulsion Flows." AEDC-TR-88-32, January 1989.

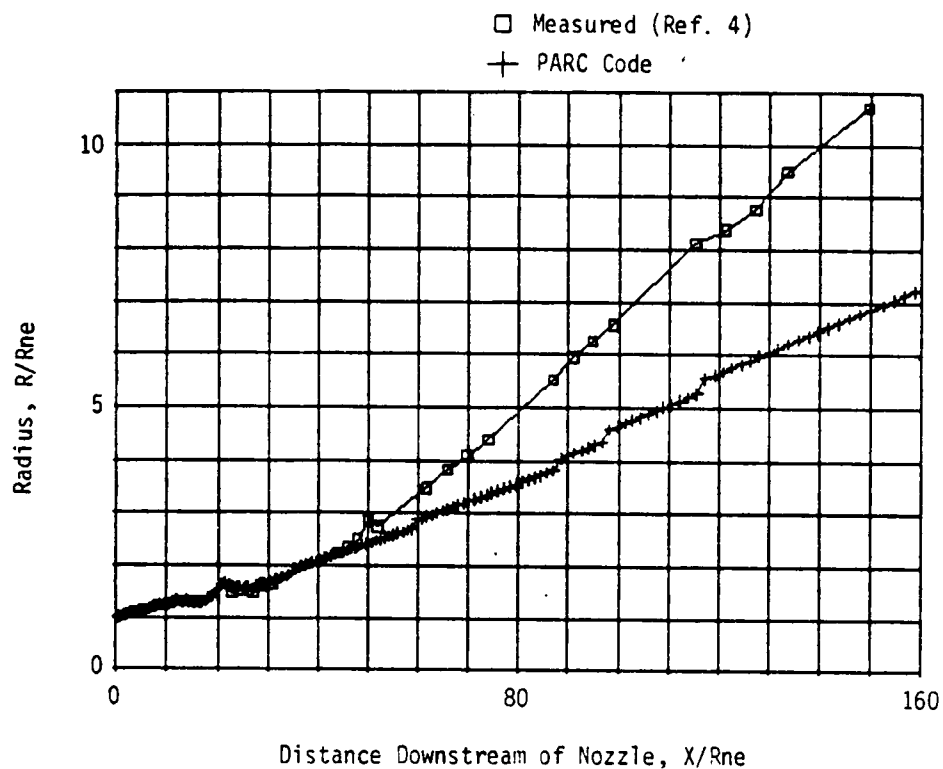


Figure 31. Far field results for supersonic nozzle [16]

Source: Cooper, G. K., Garrard, G. D., and Phares, W. J. "PARC Code Validation for Propulsion Flows." AEDC-TR-88-32, January 1989.

$H = 3.79 \text{ cm}$ $M = 0.03195$
 $ReH = 28,000$ $P_r = 15.033 \text{ psi}$
 $\dot{q}_w'' = 130 \text{ W/m}^2$ $T_r = 531^\circ R$

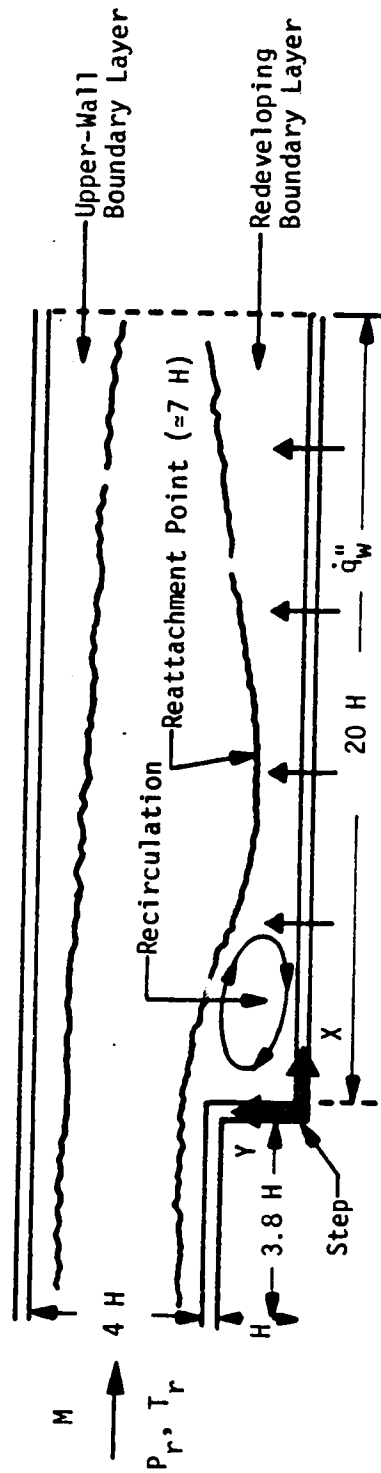
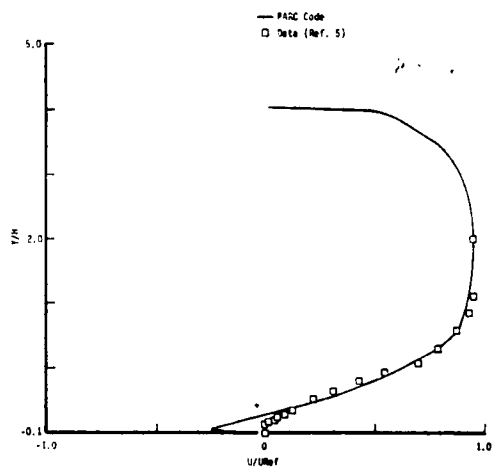
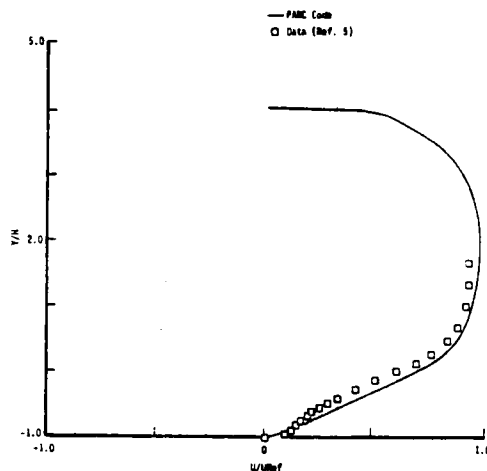


Figure 32. Rearward facing step basic formulation

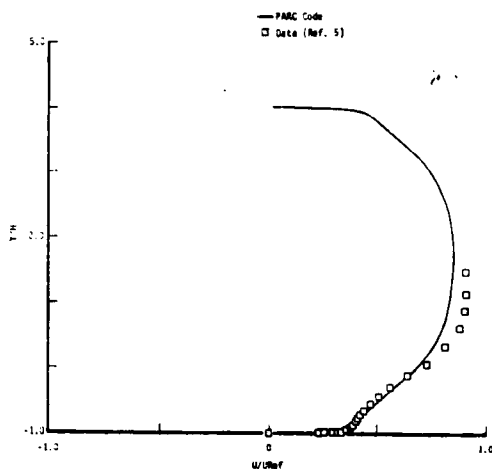
Source: Cooper, G. K., Garrard, G. D., and Phares, W. J. "PARC Code Validation for Propulsion Flows." AEDC-TR-88-32, January 1989.



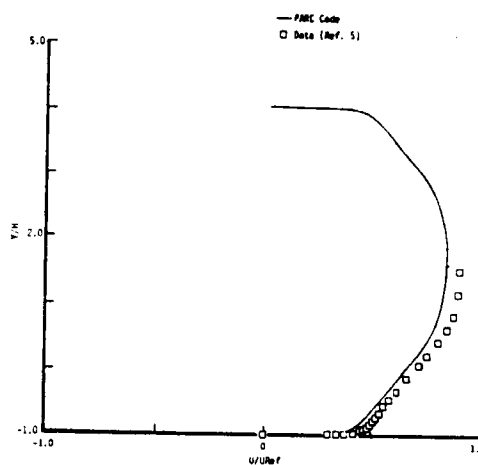
a. $X/H = 5.9$



b. $X/H = 7.2$



c. $X/H = 12.2$

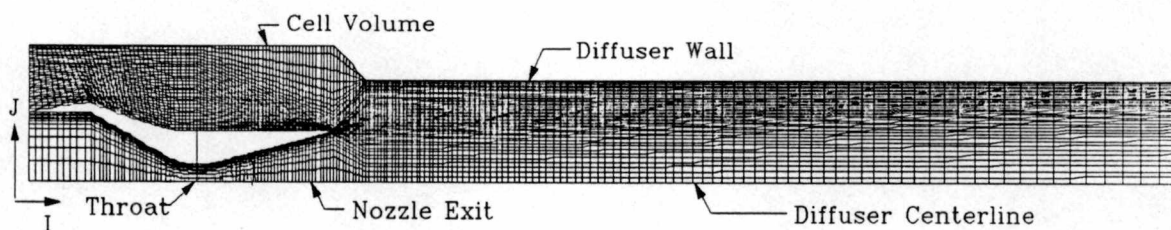


d. $X/H = 17.5$

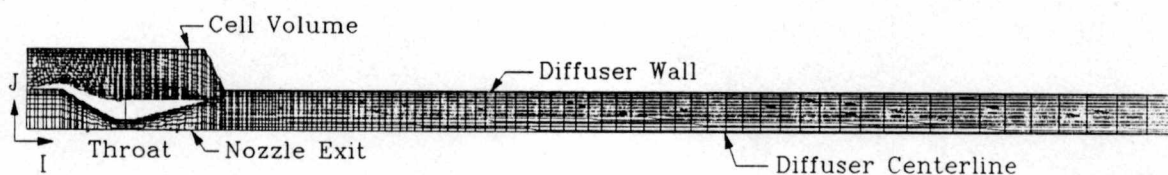
Figure 33. Velocity profiles downstream of rearward facing step

Source: Cooper, G. K., Garrard, G. D., and Phares, W. J. "PARC Code Validation for Propulsion Flows." AEDC-TR-88-32, January 1989.

SOUTHWORTH CO. U.S.A.
100% COTTON FIBER



a. Diffuser 1 grid - 1.5 inch diameter,
6 inch length



b. Diffuser 2 grid - 1 inch diameter,
12 inch length

Figure 34. Gridding used in the calibration of the PARC2D
program for steady state performance predictions (every
other J indice shown)

PARCIMENT DEED

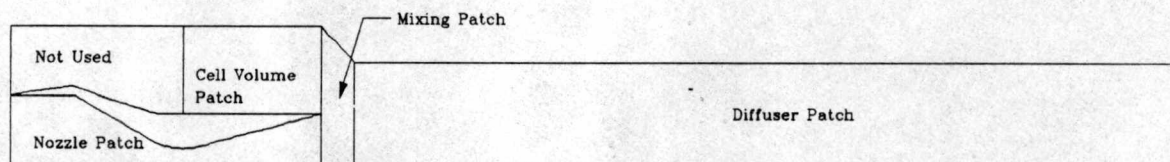


Figure 35. Grid patches (Diffuser 1 shown, Diffuser 2 Similar)

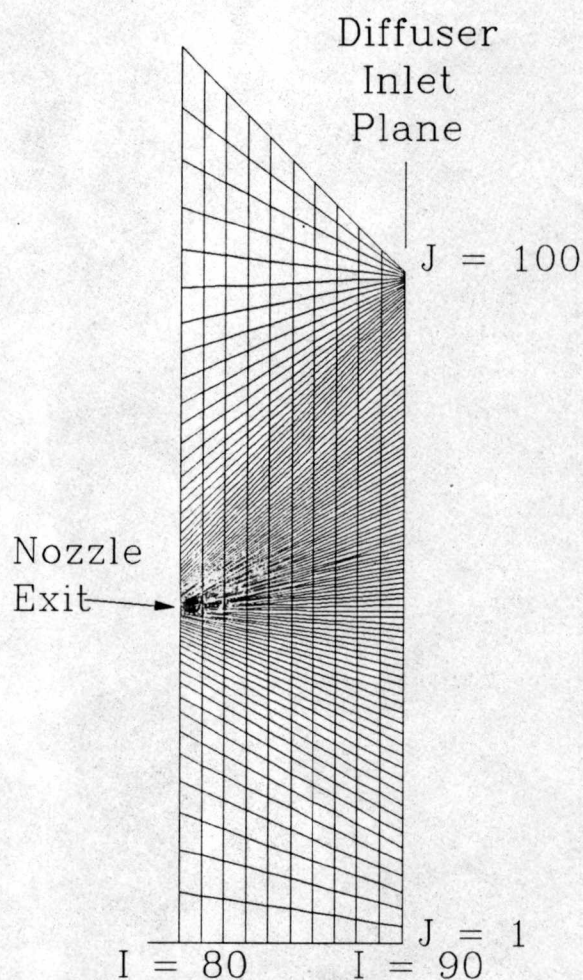


Figure 36. Mixing patch grid resolution

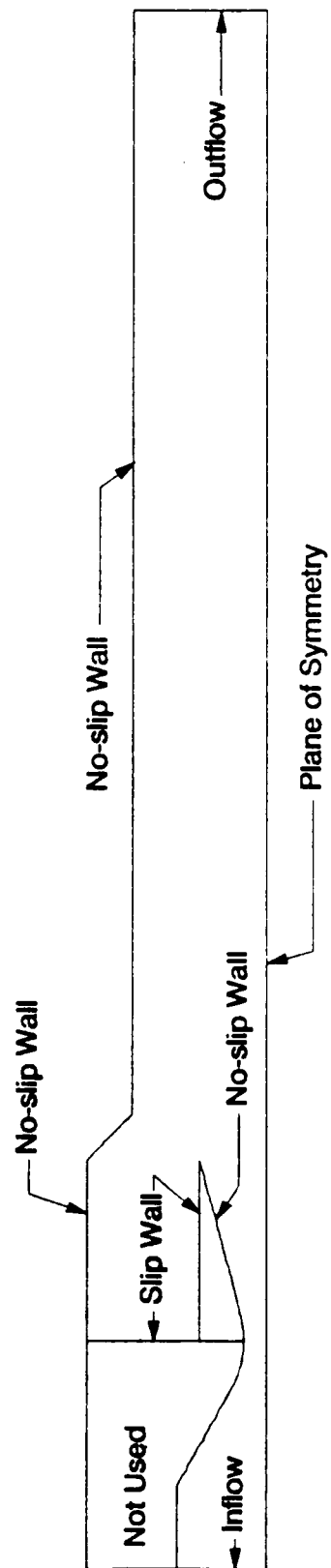
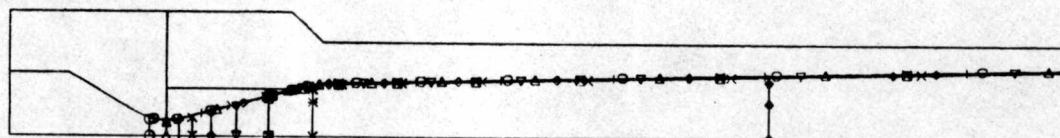
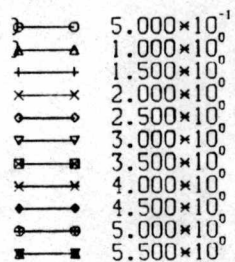
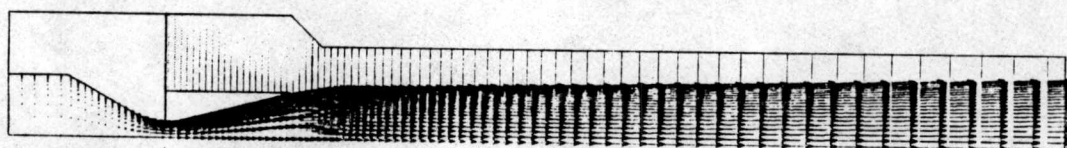


Figure 37. Boundary conditions applied over the computational domain



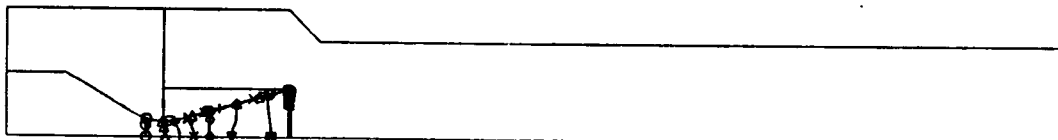
a. Mach number contours



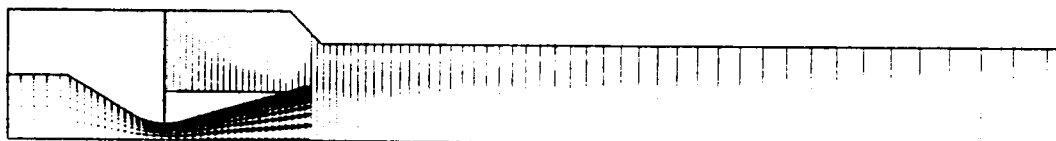
b. Velocity vectors

Figure 38. Started diffuser initial conditions

○—○	5.000×10^{-1}
△—△	1.000×10^0
†—†	1.500×10^0
×—×	2.000×10^0
◇—◇	2.500×10^0
▽—▽	3.000×10^0
■—■	3.500×10^0
×—×	4.000×10^0
●—●	4.500×10^0
●—●	5.000×10^0
■—■	5.500×10^0

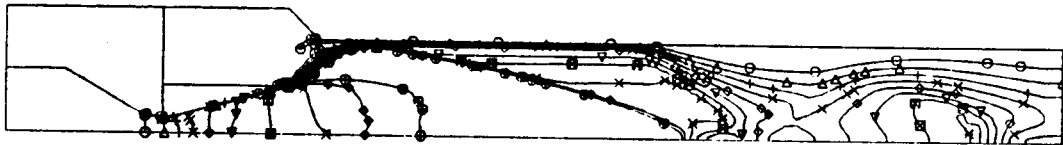
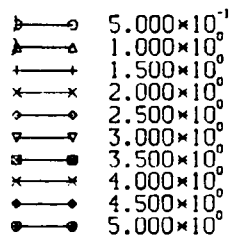


a. Mach number contours

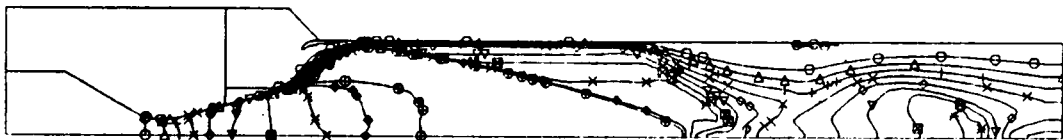


b. Velocity vectors

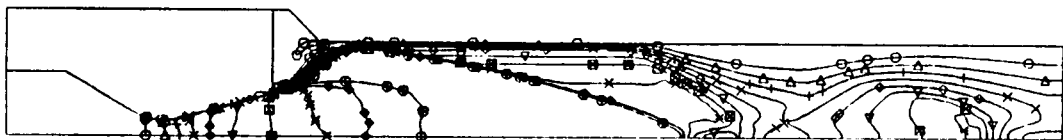
Figure 39. Unstarted diffuser initial conditions



a. Full cell volume

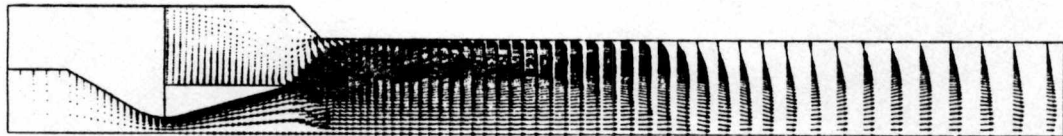


b. Half cell volume

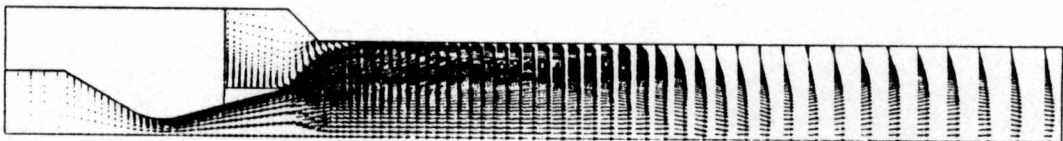


c. No cell volume

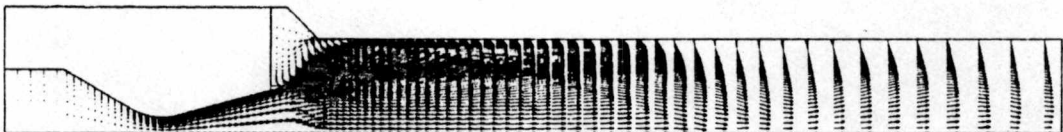
Figure 40. Mach number contours for the cell volume parametric study



a. Full cell volume



b. Half cell volume



c. No cell volume

Figure 41. Velocity vectors for the cell volume parametric study

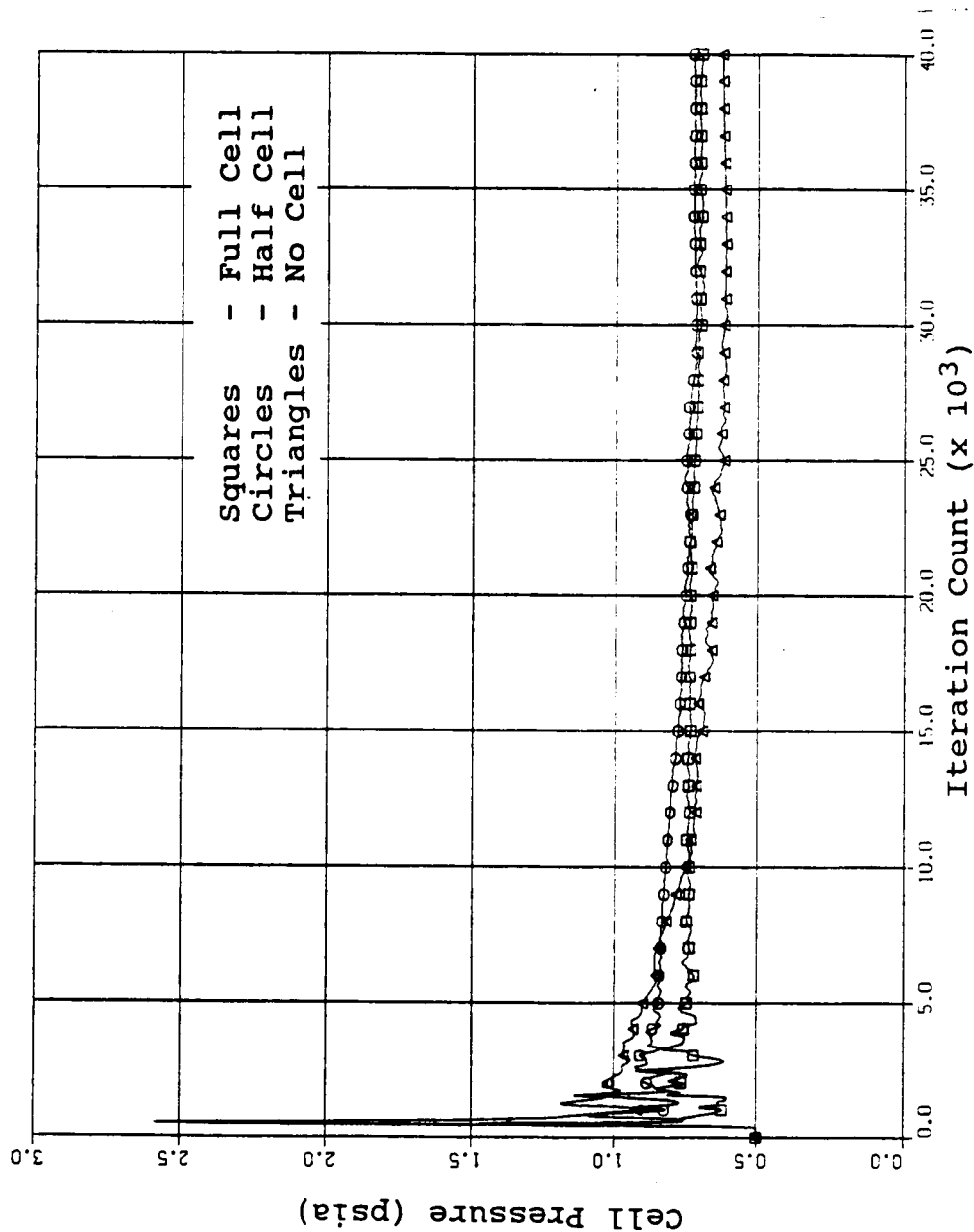
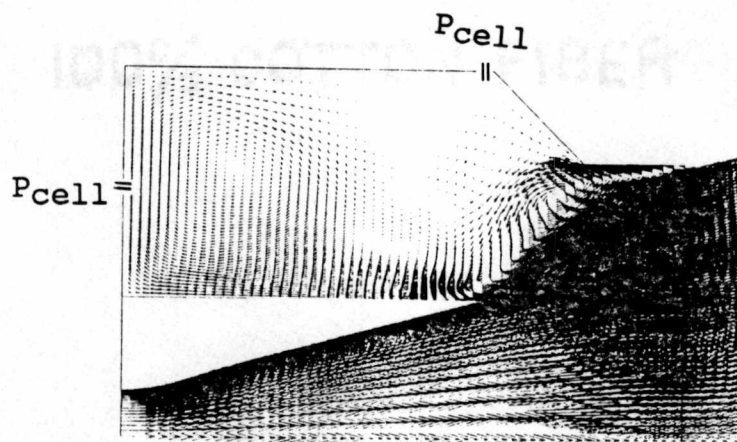
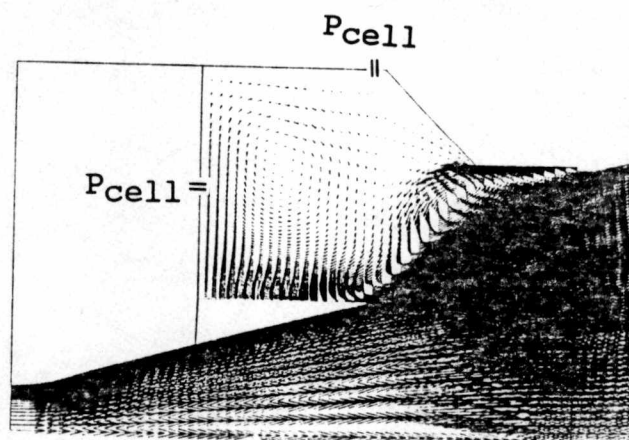


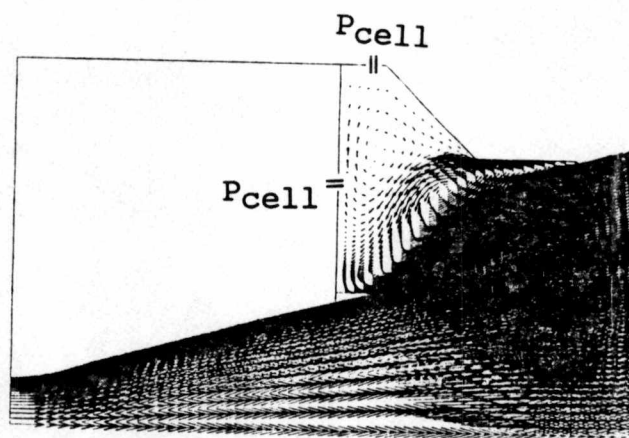
Figure 42. Cell pressure as a function of iteration count for the cell volume parametric study



a. Full cell volume



b. Half cell volume



c. No cell volume

Figure 43. Flow field in the cell volume for the cell volume parametric study

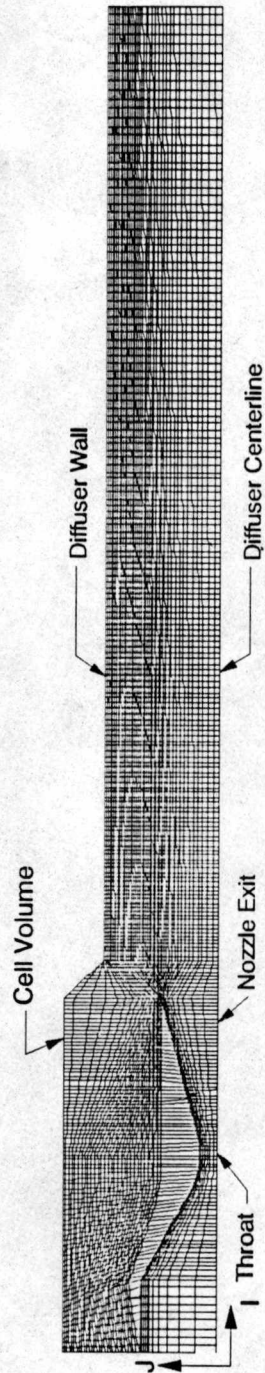
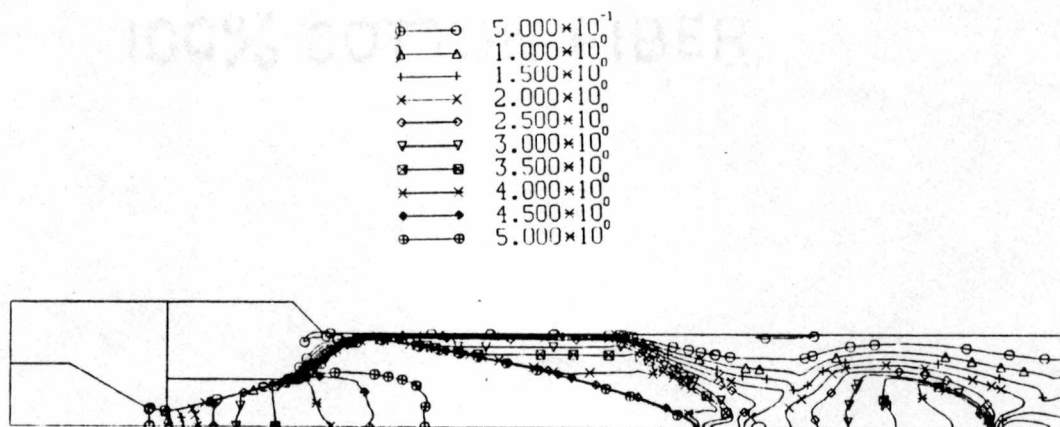


Figure 44. 240 x 100 grid for the diffuser gridding parametric study

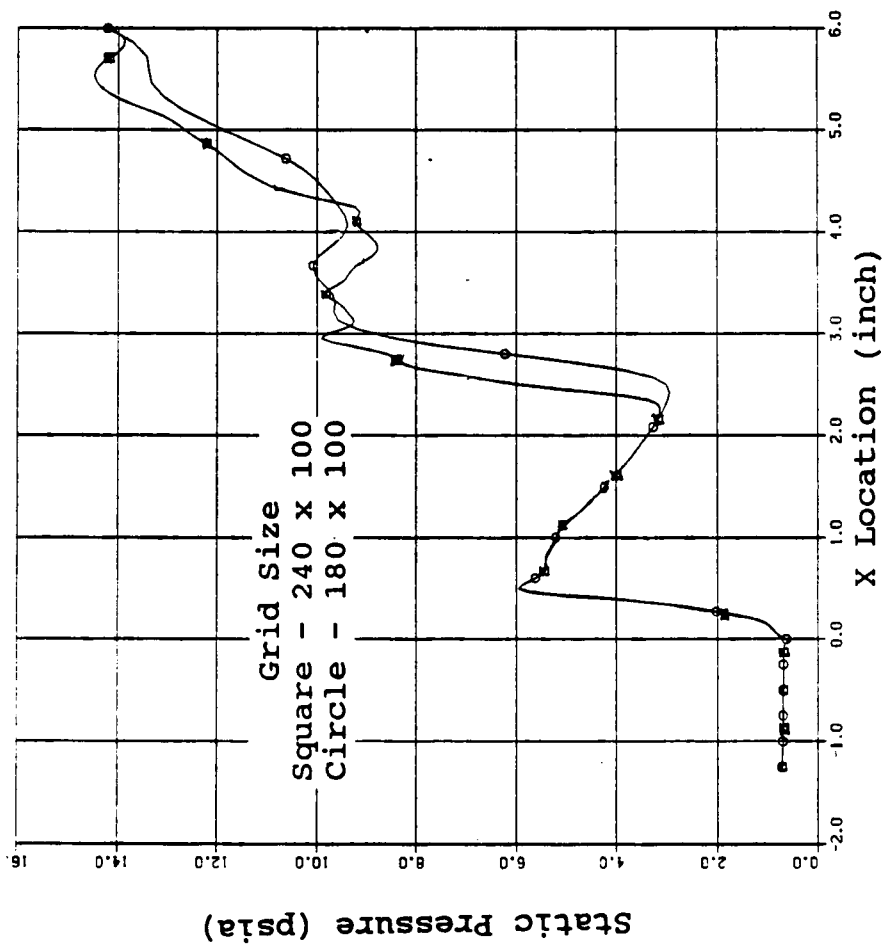


a. Mach number contours



b. Velocity vectors

Figure 45. Results from using the 240 x 100 grid in the $L/D = 4$ diffuser



c. Diffuser wall static pressure comparison

Figure 45. Results from using the 240 x 100 grid in the $L/D = 4$ diffuser (concluded)

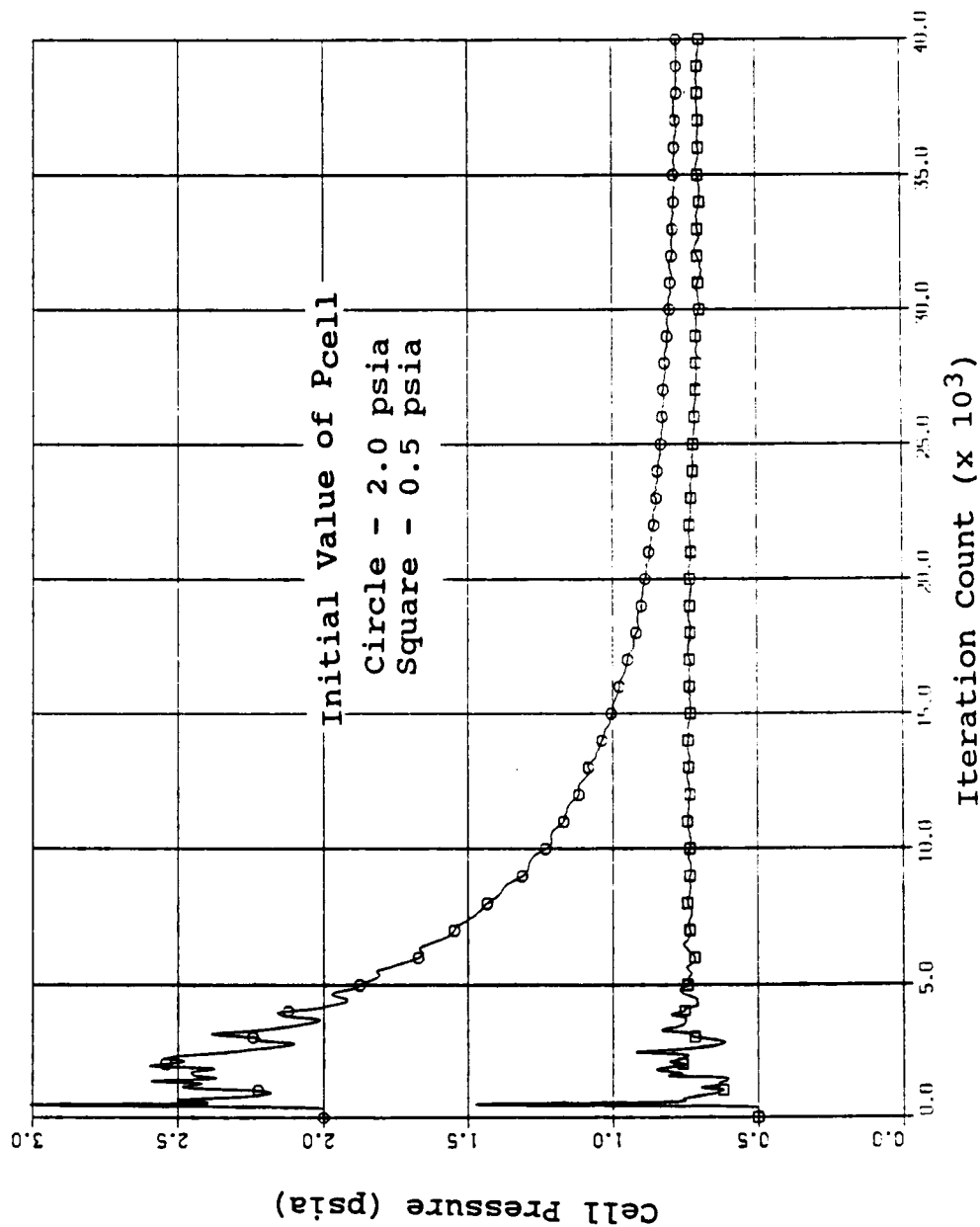


Figure 46. Cell pressure as a function of iteration count comparison for the initial cell volume pressure parametric study

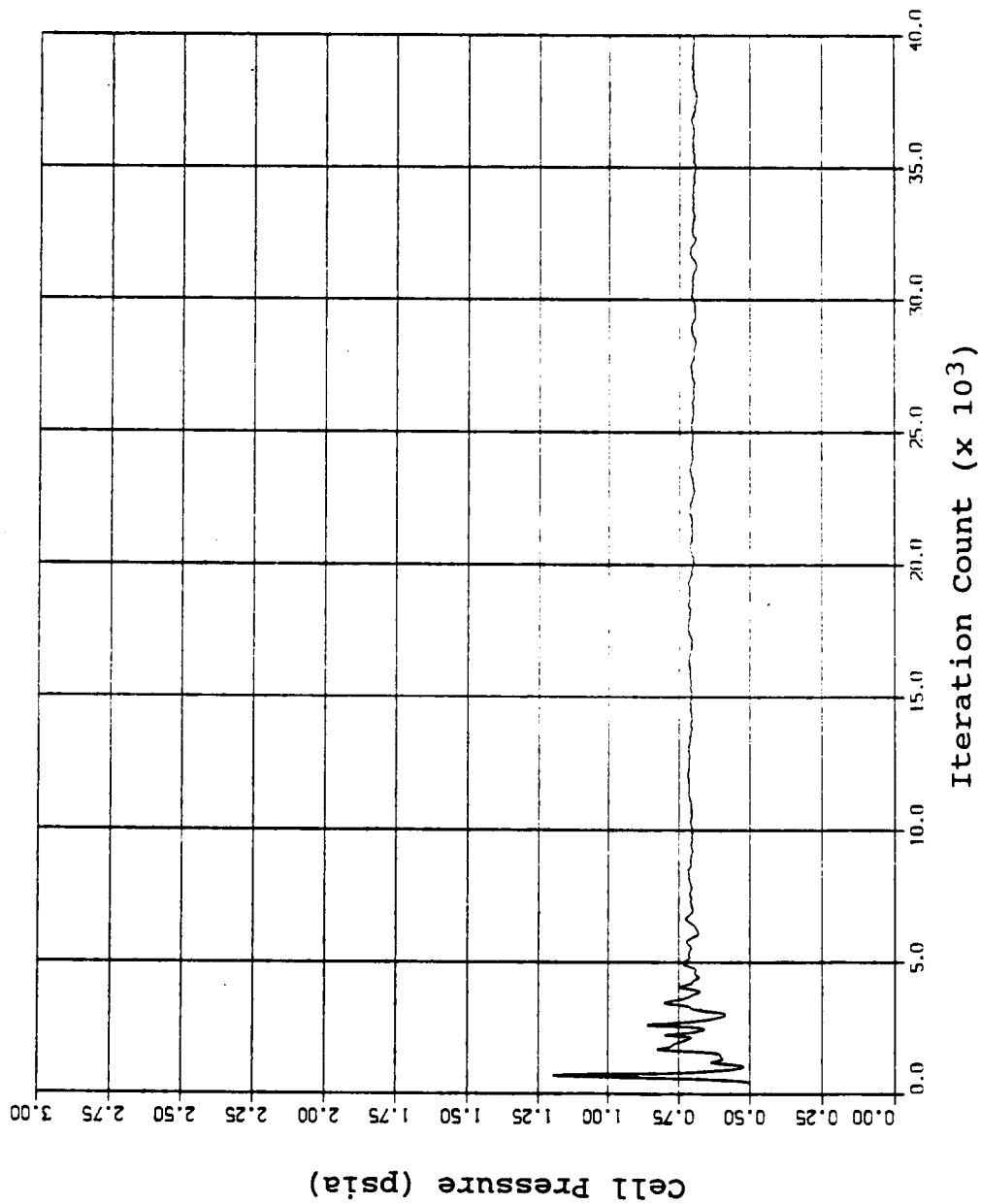
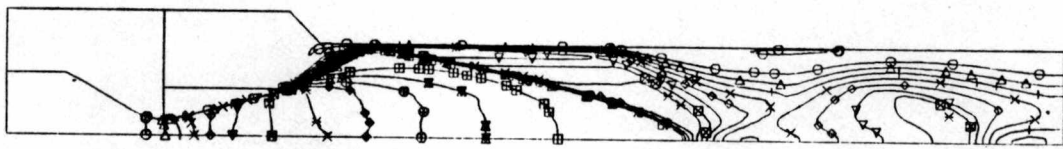
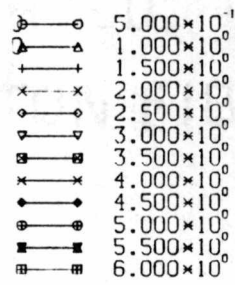
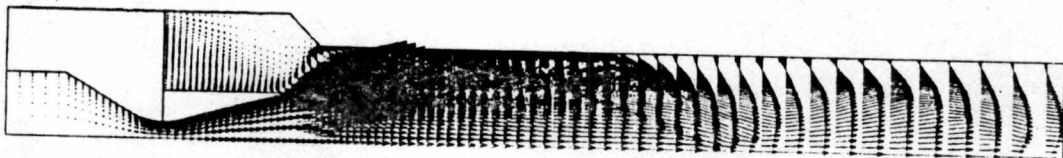


Figure 47. Cell pressure as a function of iteration count started with the unstated diffuser initial conditions.

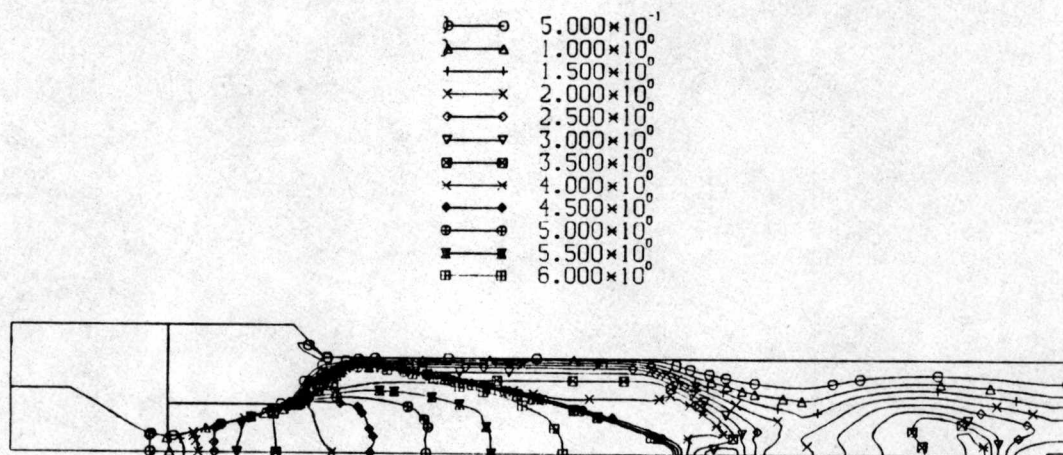


a. Mach number contours

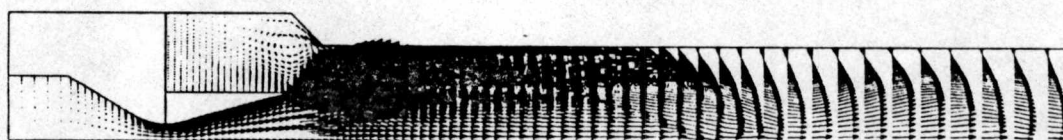


b. Velocity vectors

Figure 48. Flow solution for COFMIX = 0.09



a. Mach number contours



b. Velocity vectors

Figure 49. Flow solution for COFMIX = 0.13

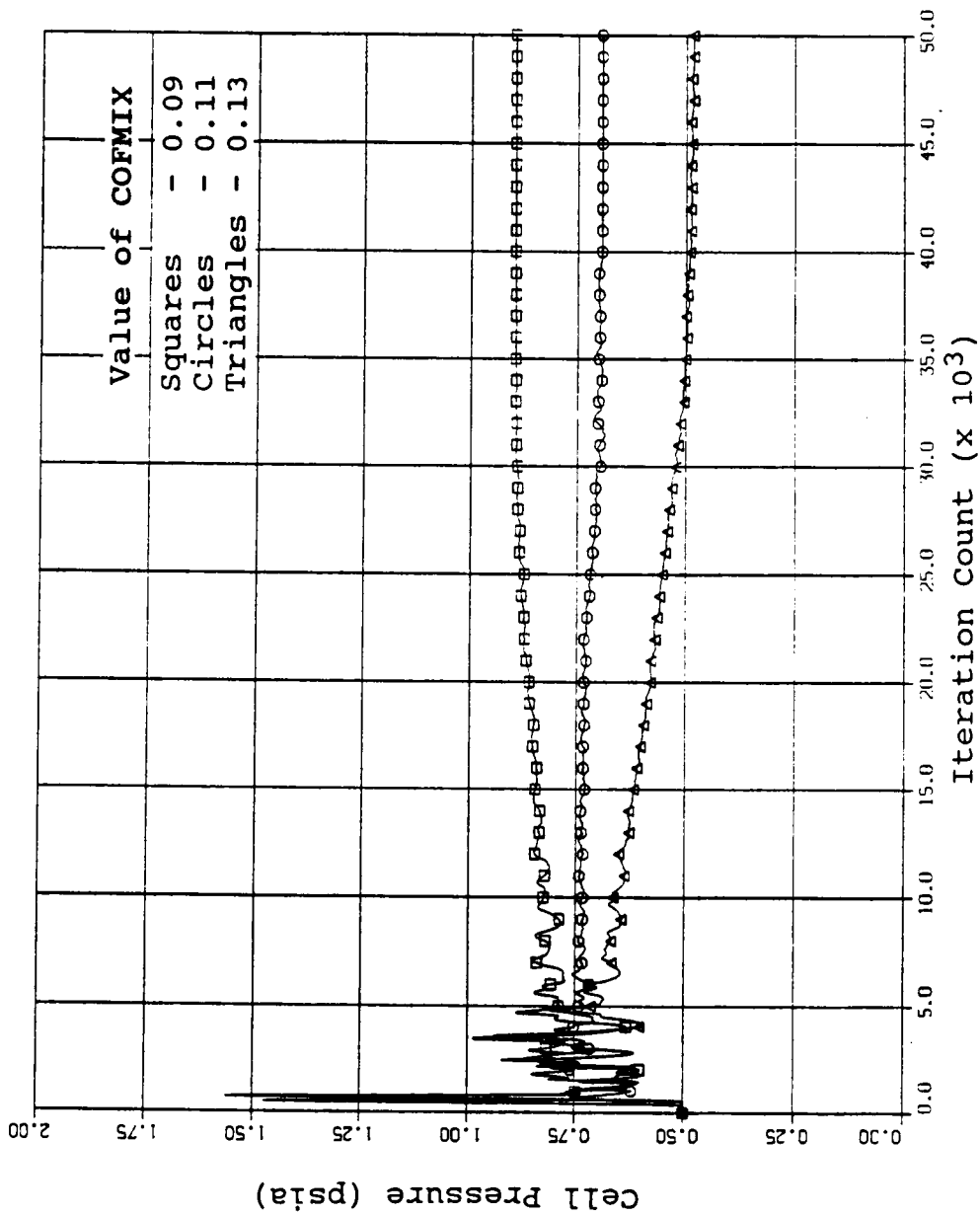


Figure 50. Cell pressure as a function of iteration count for the turbulence mixing coefficient parametric study

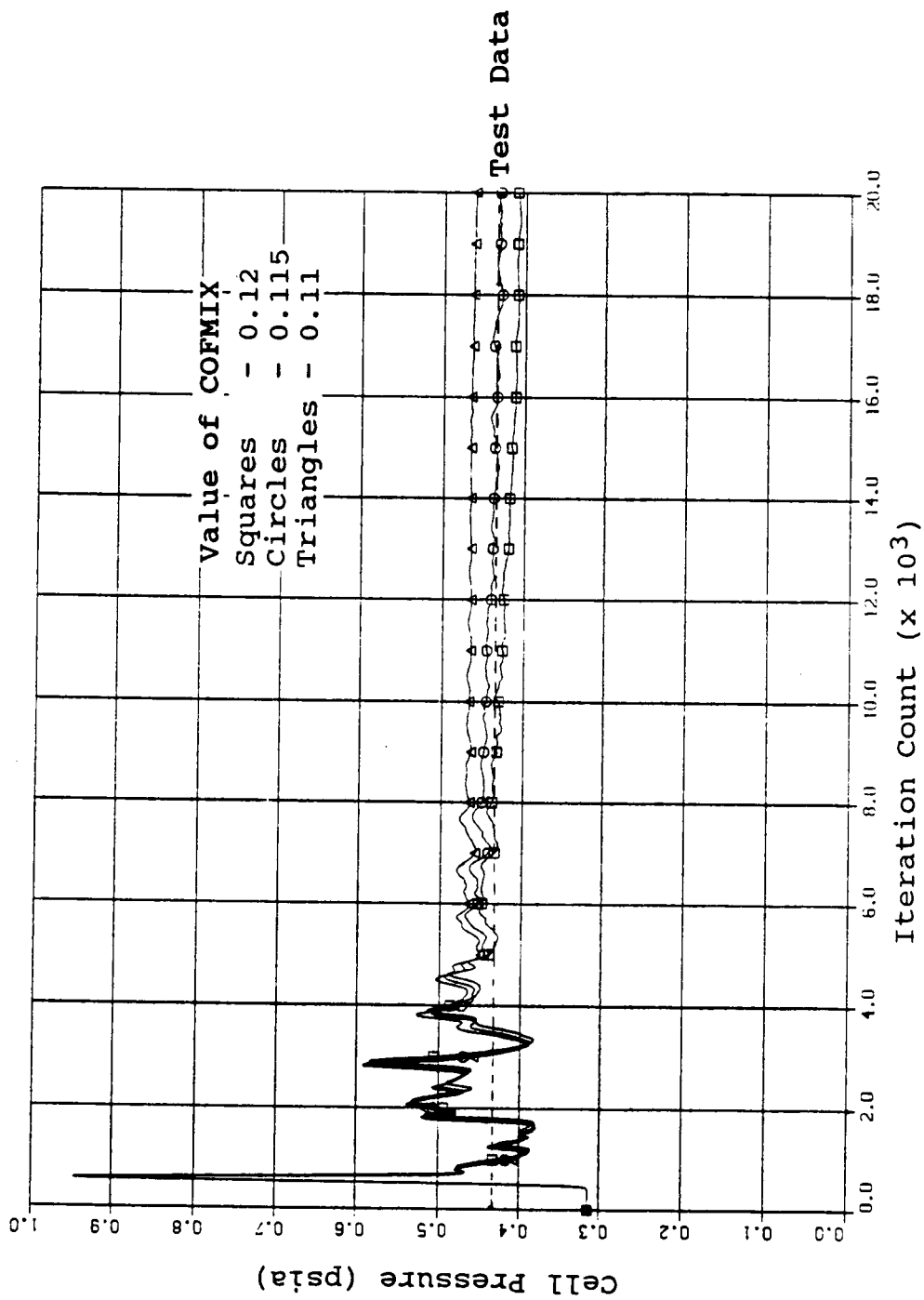
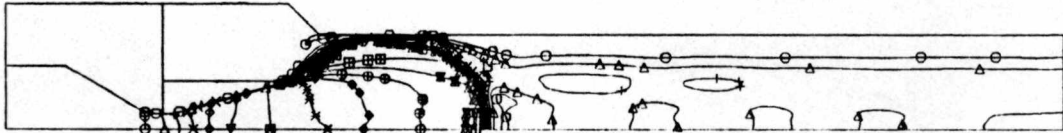
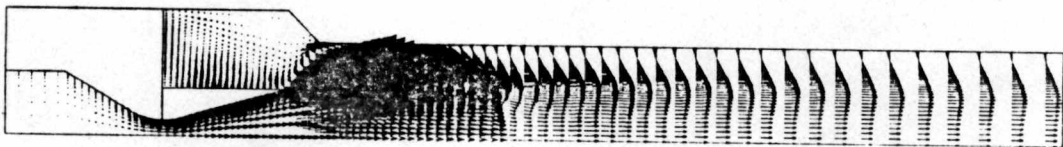


Figure 51. Cell pressure as a function of iteration count for Case 1

○	5.000×10^{-1}
△	1.000×10^0
+	1.500×10^0
×	2.000×10^0
◇	2.500×10^0
▽	3.000×10^0
□	3.500×10^0
×	4.000×10^0
◇	4.500×10^0
□	5.000×10^0
×	5.500×10^0
□	6.000×10^0



a. Mach number contours



b. Velocity vectors

Figure 52. Flow solution for Case 1

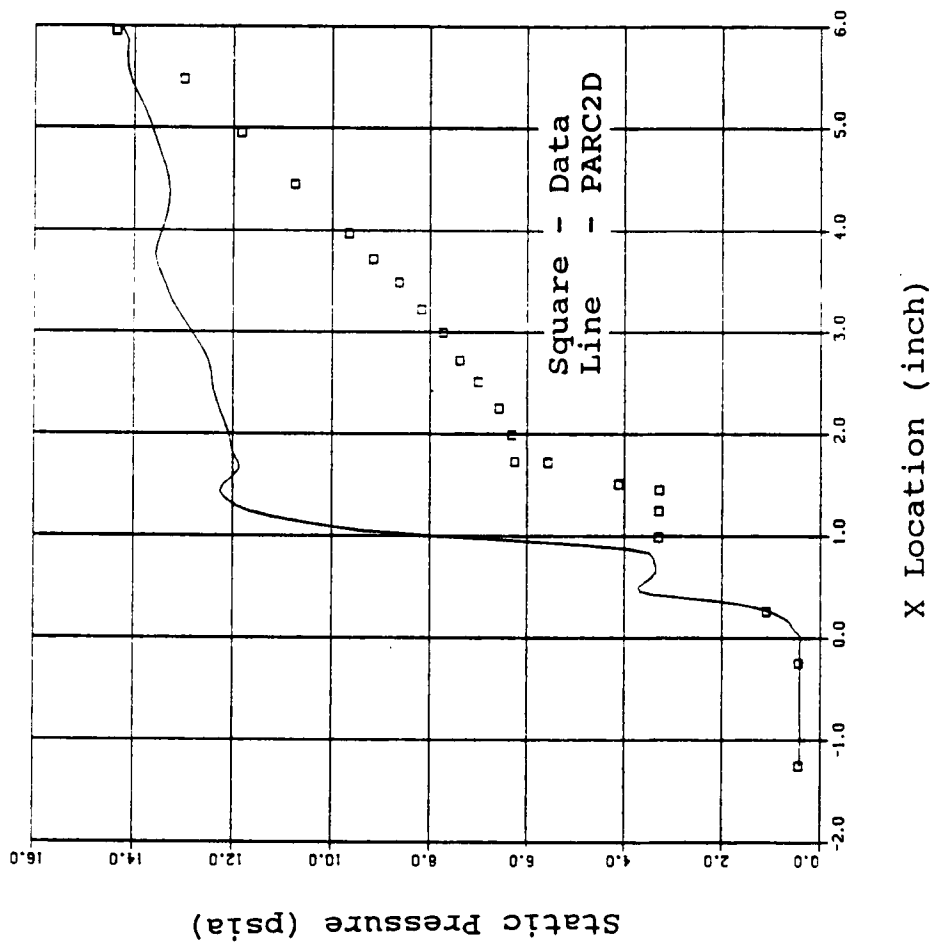


Figure 53. Diffuser wall static pressure comparison for Case 1

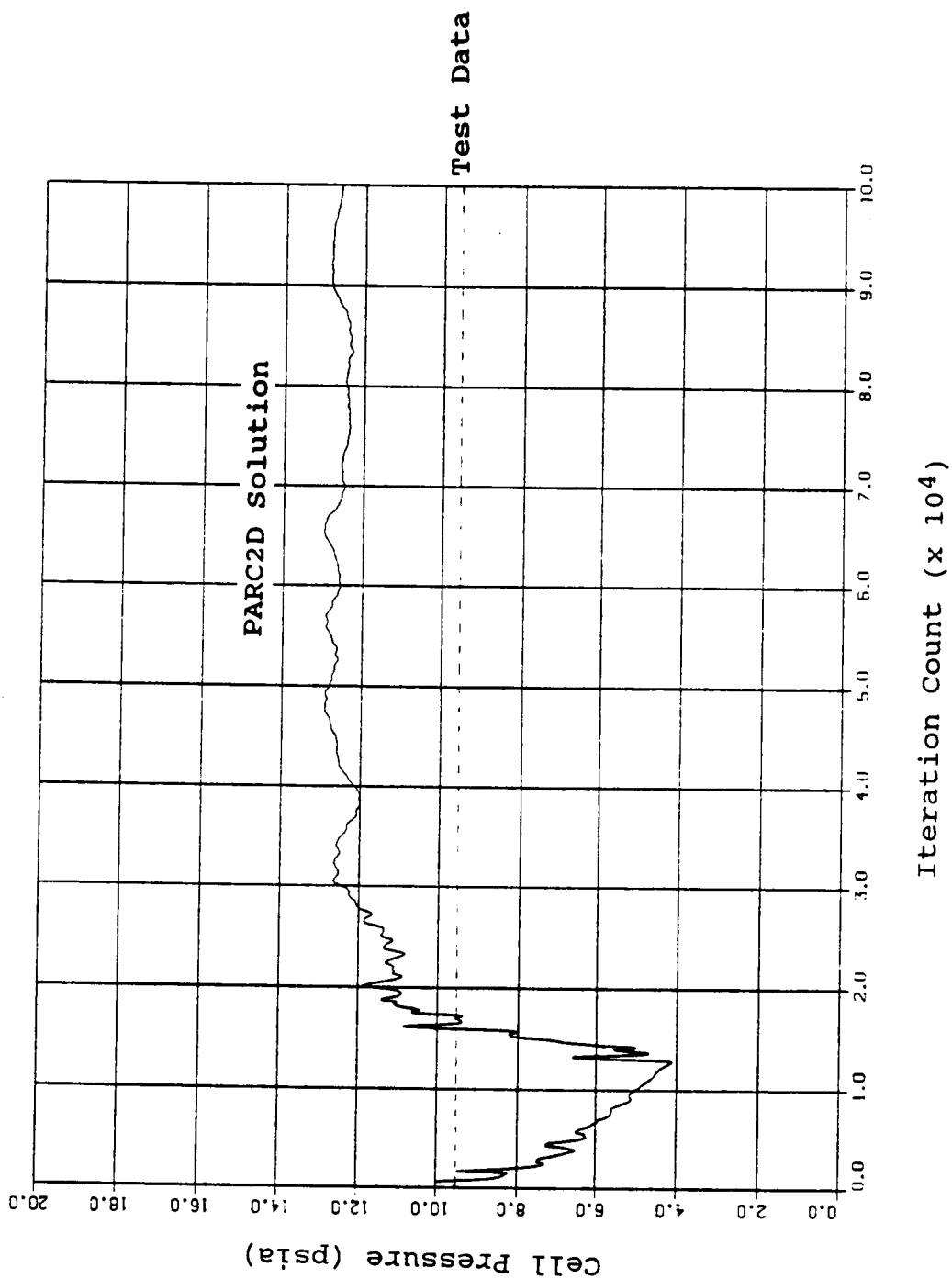
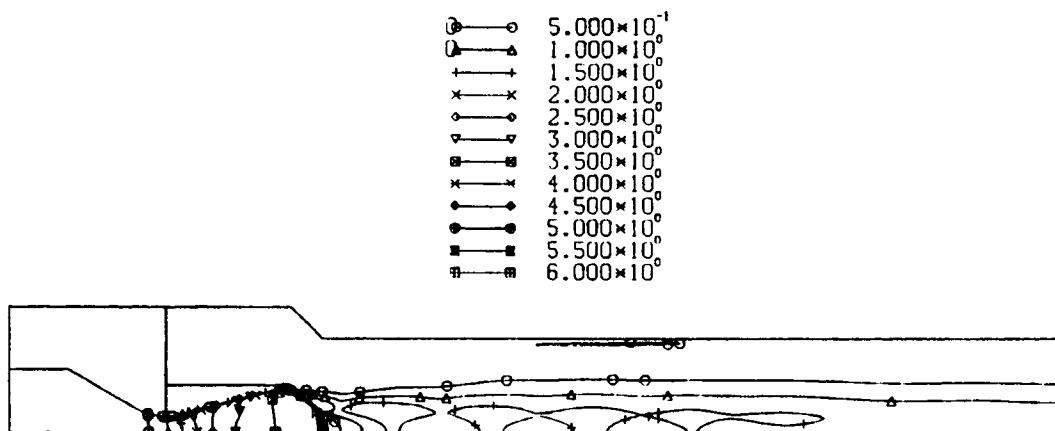
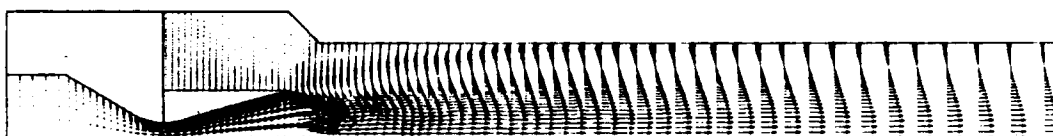


Figure 54. Cell pressure as a function of iteration count for Case 2

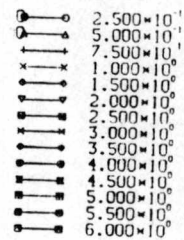
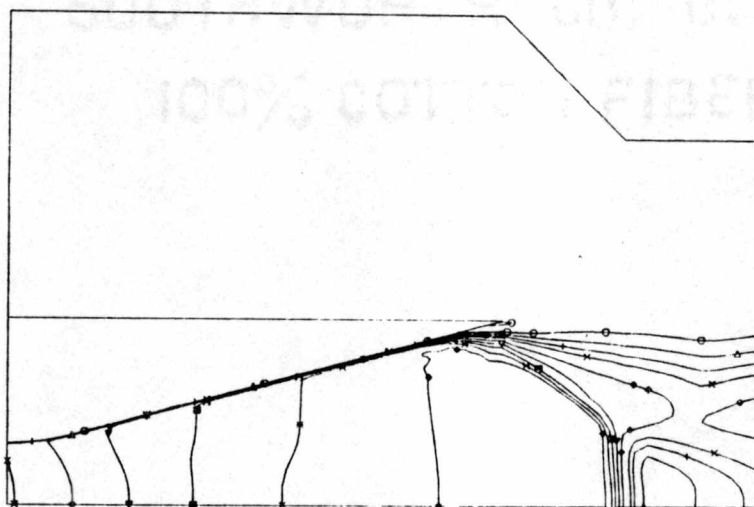


a. Mach number contours

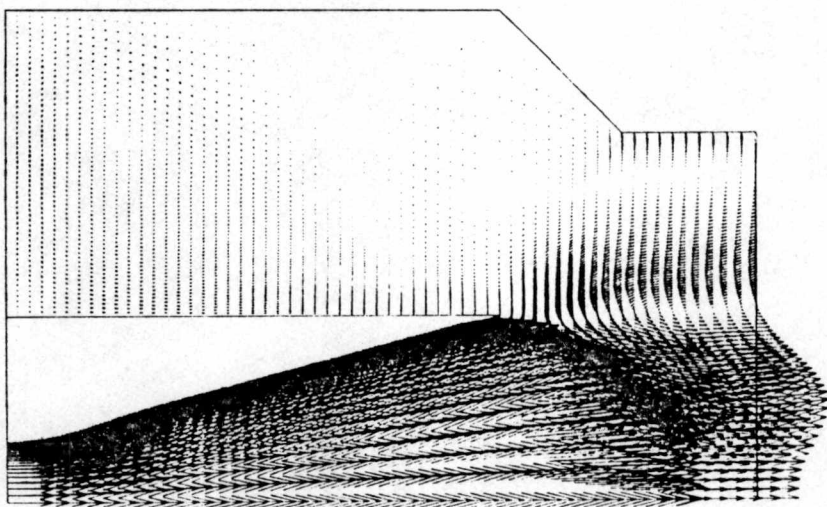


b. Velocity vectors

Figure 55. Flow solution for Case 2



a. Mach number contours



b. Velocity vectors

Figure 56. Nozzle flow solution for Case 2

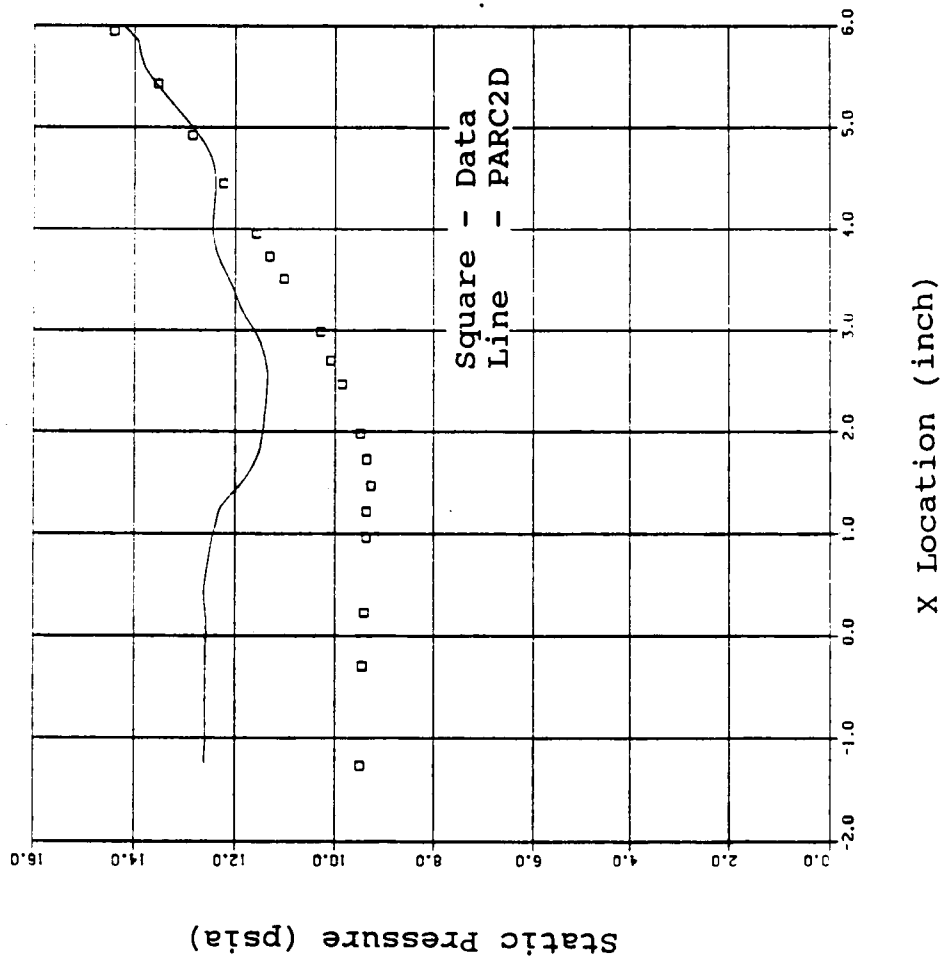


Figure 57. Diffuser wall static pressure comparison for Case 2

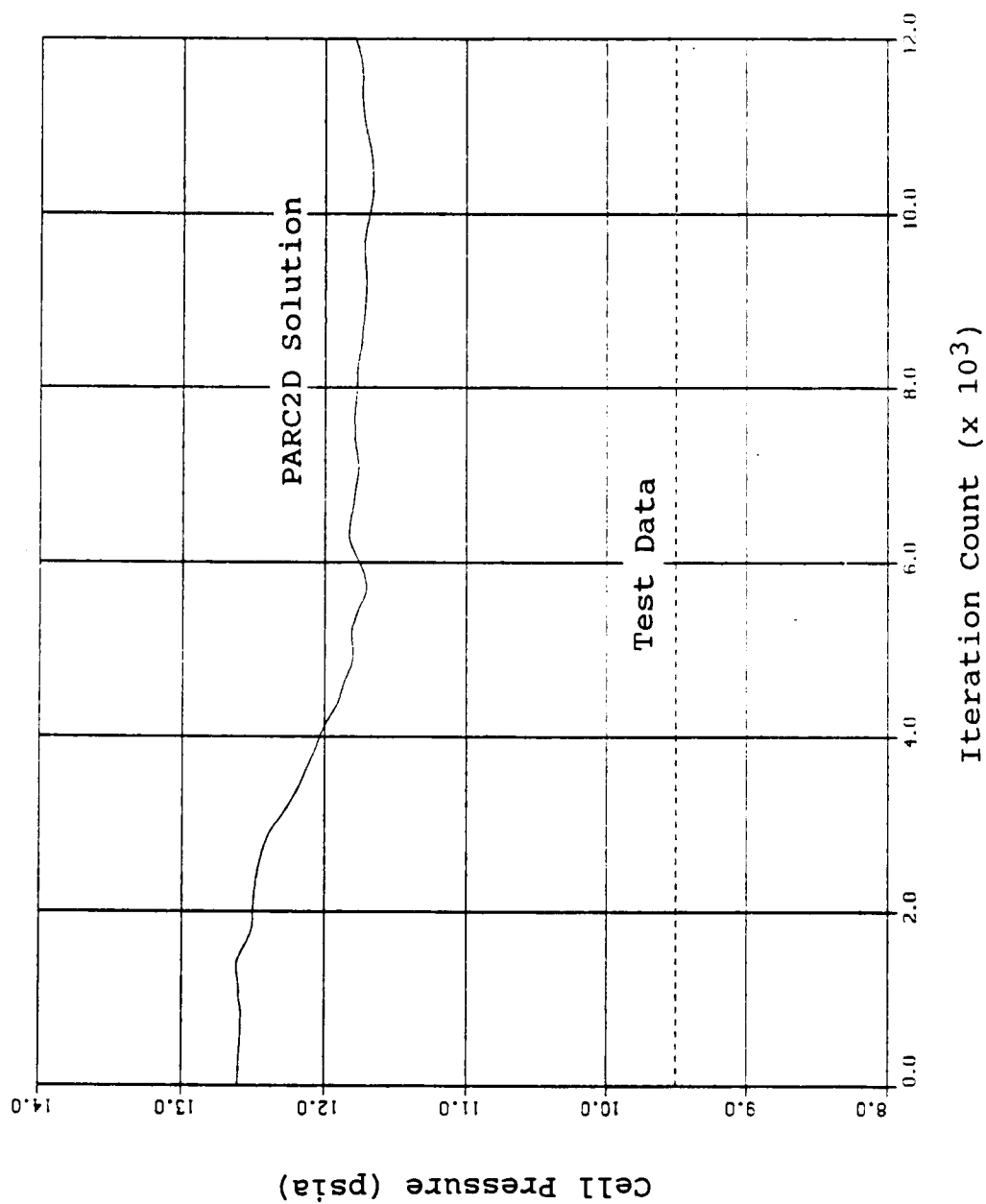


Figure 58. Cell pressure as a function of iteration count for Case 2 with COFMIX = 0.13

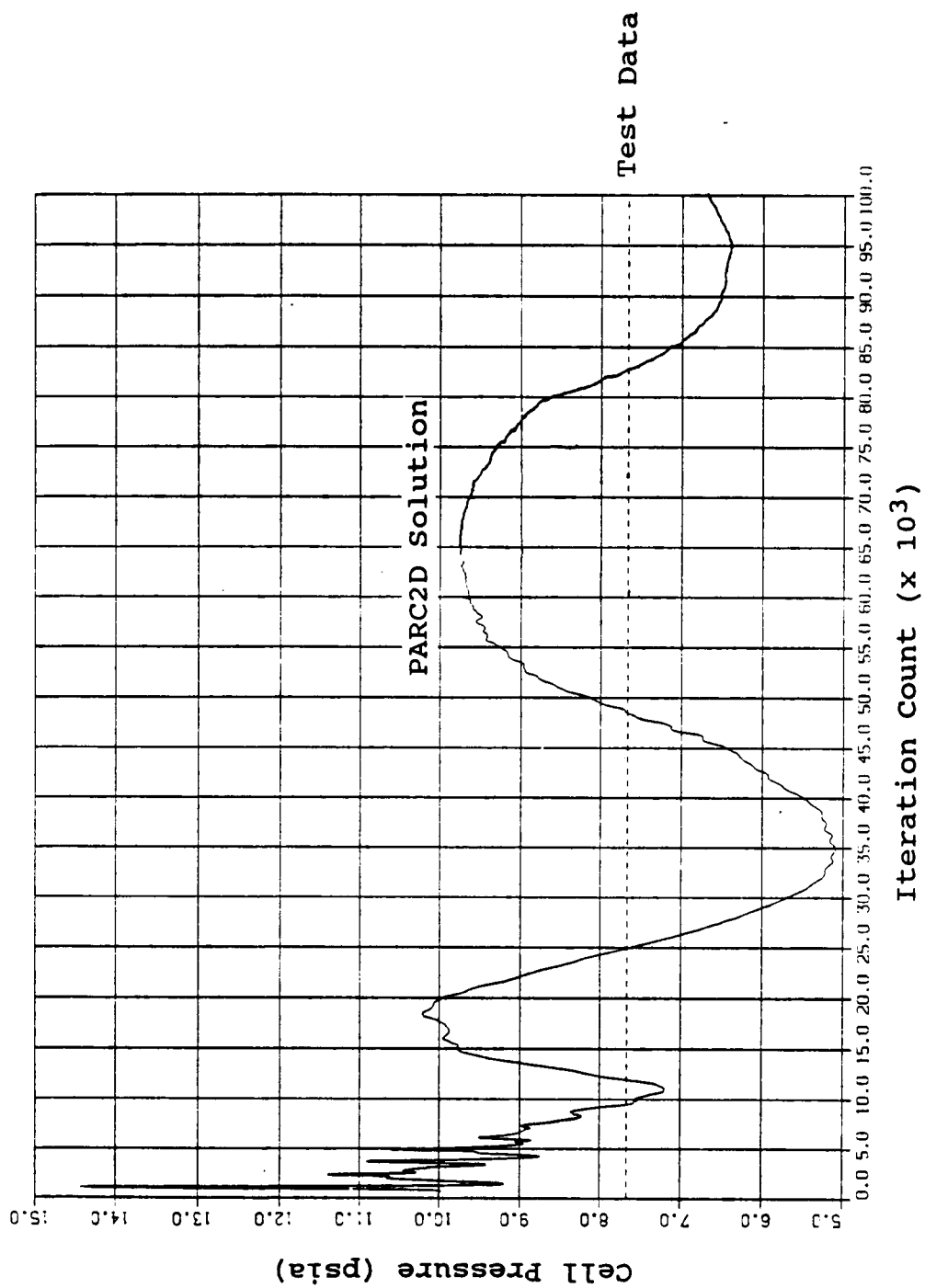
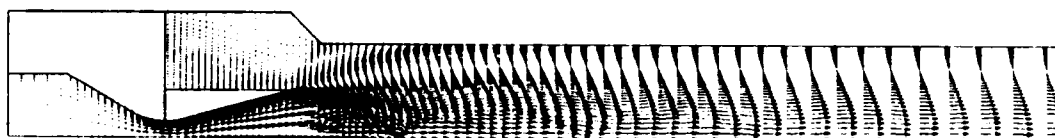


Figure 59. Cell pressure as a function of iteration count for Case 3

○—○	5.000×10^{-1}
△—△	1.000×10^0
+—+	1.500×10^0
×—×	2.000×10^0
◇—◇	2.500×10^0
▽—▽	3.000×10^0
■—■	3.500×10^0
×—×	4.000×10^0
●—●	4.500×10^0
⊙—⊙	5.000×10^0
■—■	5.500×10^0
⊞—⊞	6.000×10^0



a. Mach number contours



b. Velocity vectors

Figure 60. Flow solution for Case 3

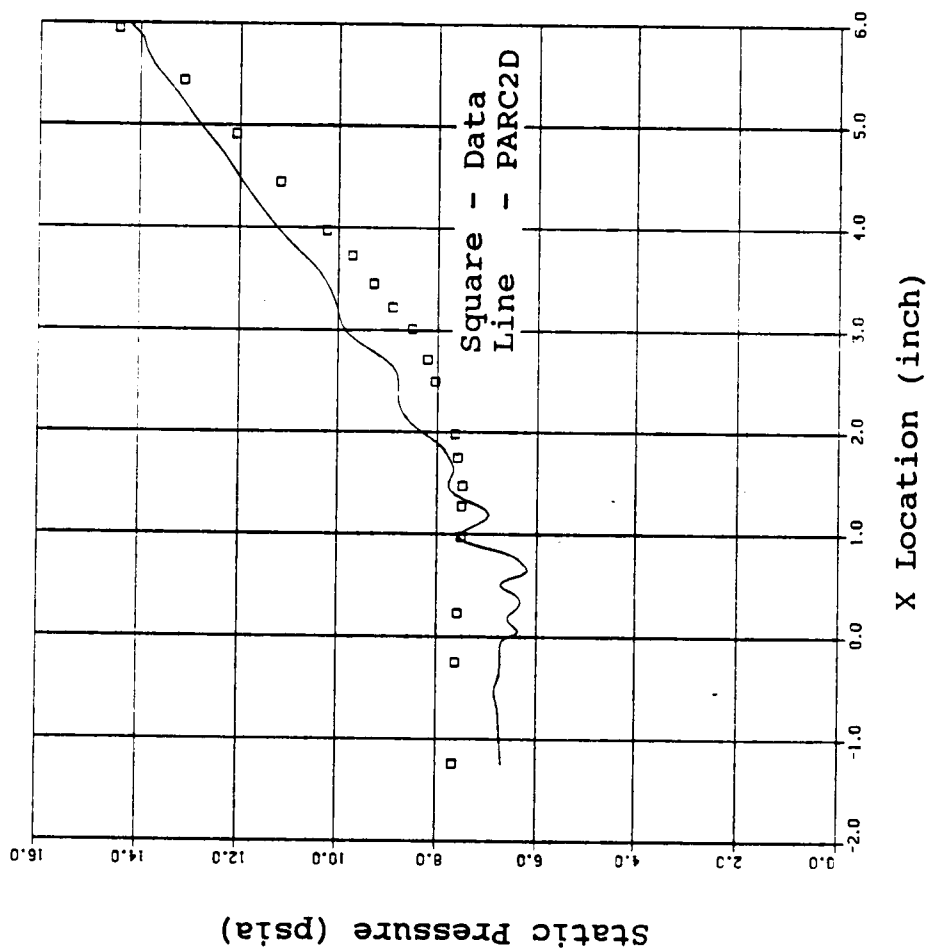


Figure 61. Diffuser wall static pressure comparison for Case 3

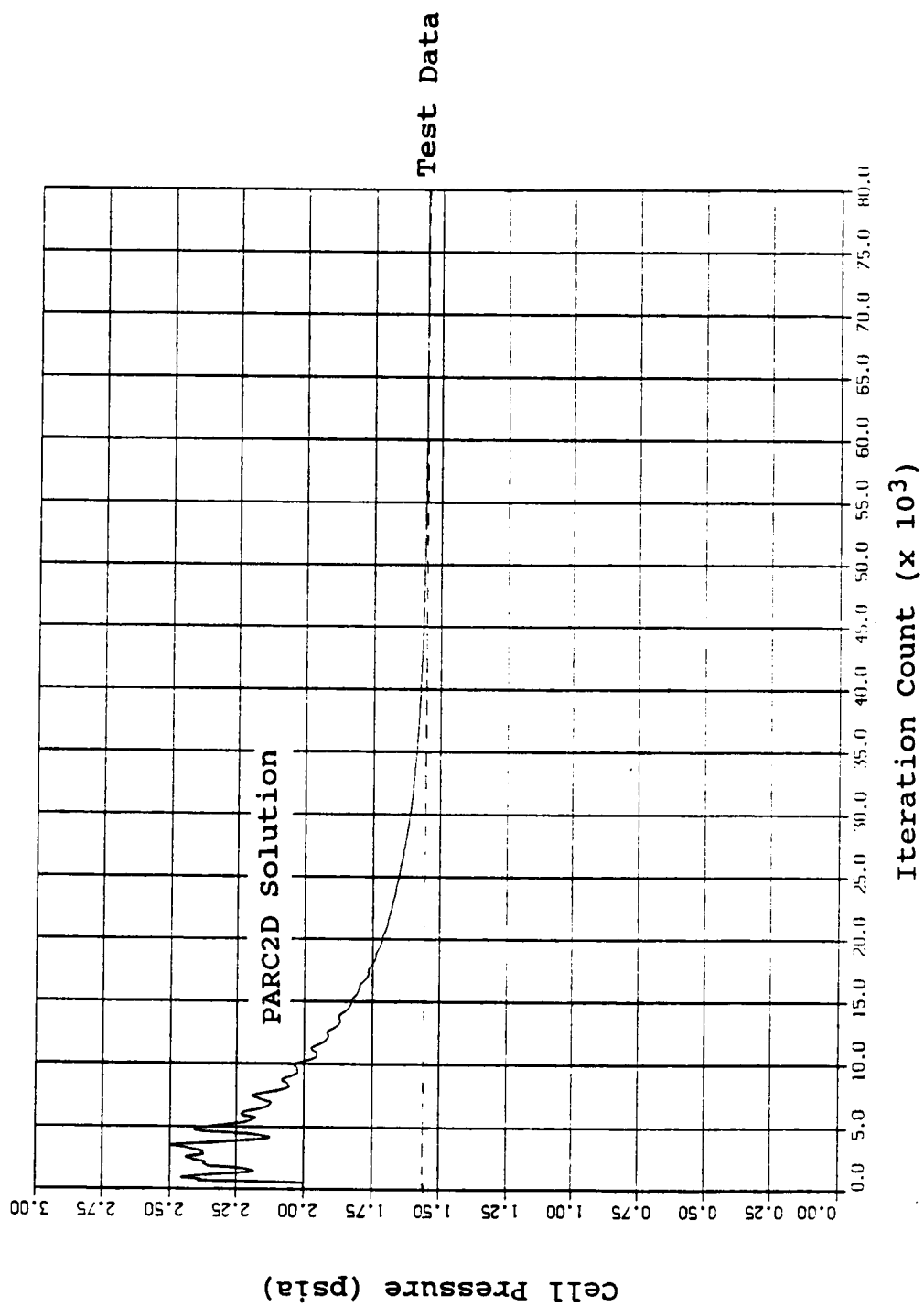
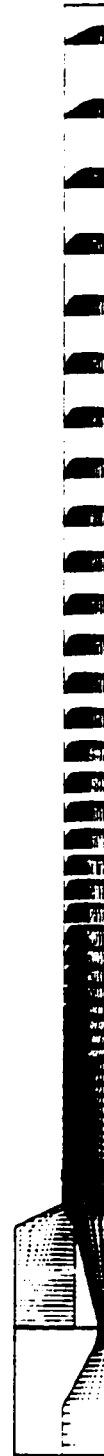


Figure 62. Cell pressure as a function of iteration count for Case 4

○	5.000×10^{-1}
△	1.000×10^0
+	1.500×10^0
x	2.000×10^0
◇	2.500×10^0
▽	3.000×10^0
□	3.500×10^0
×	4.000×10^0
◆	4.500×10^0
⊙	5.000×10^0
■	5.500×10^0
⊞	6.000×10^0



a. Mach number contours



b. Velocity vectors

Figure 63. Flow solution for Case 4

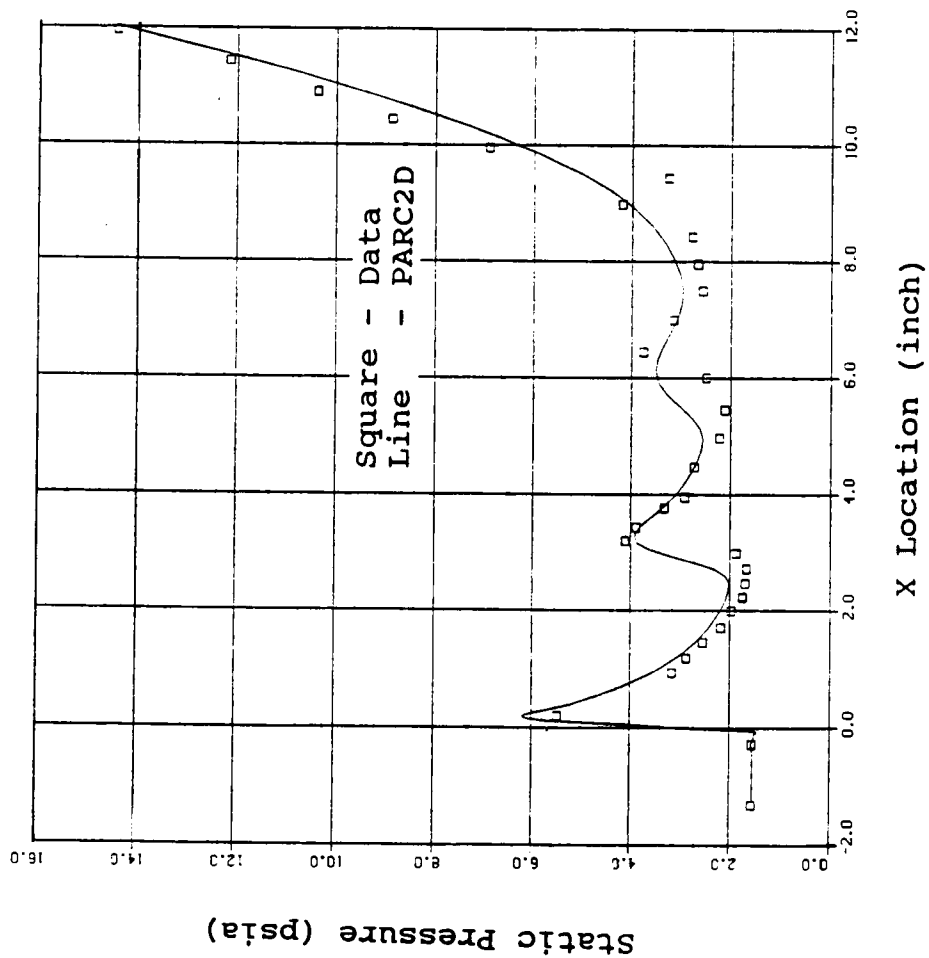


Figure 64. Diffuser wall static pressure comparison for Case 4

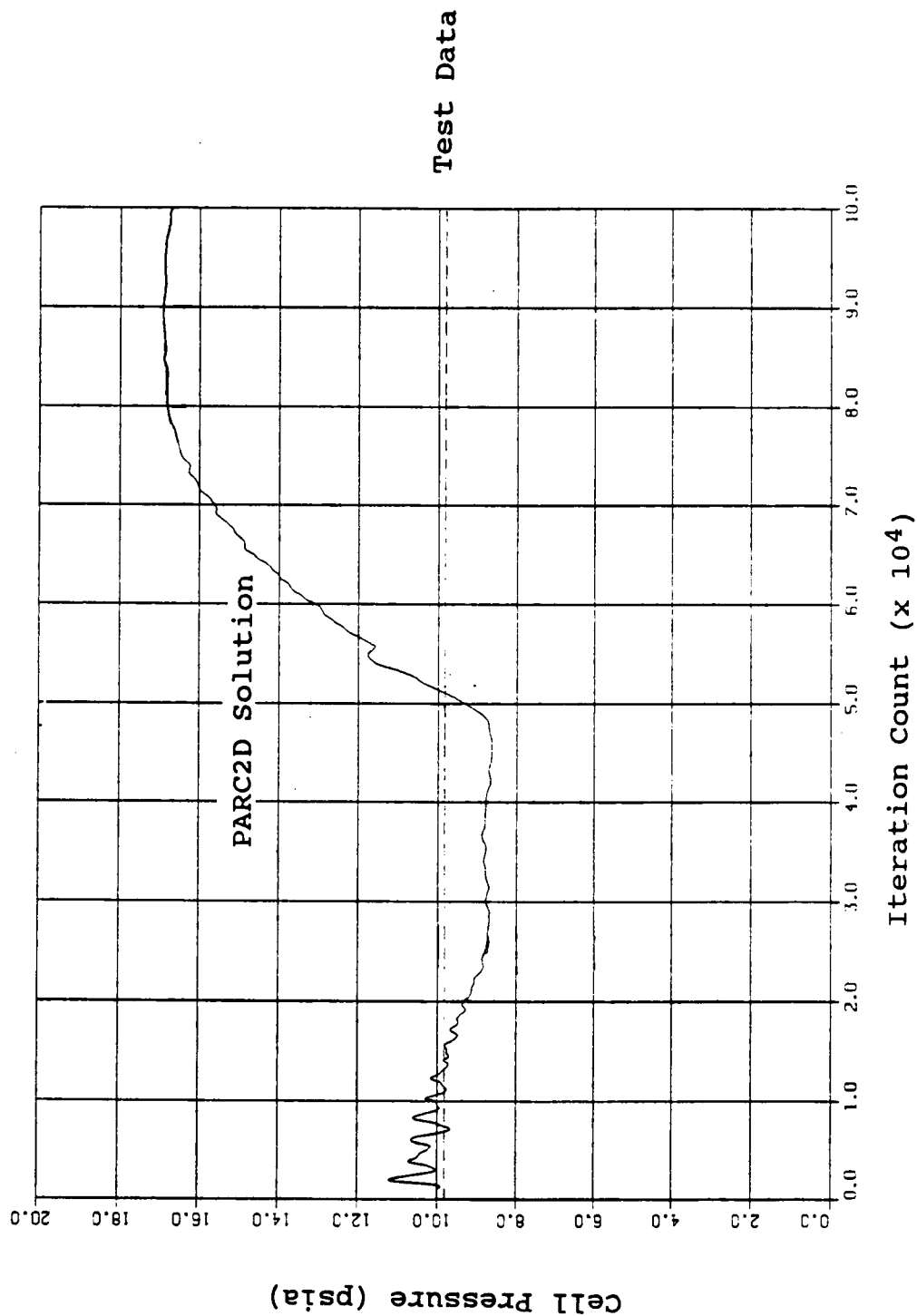
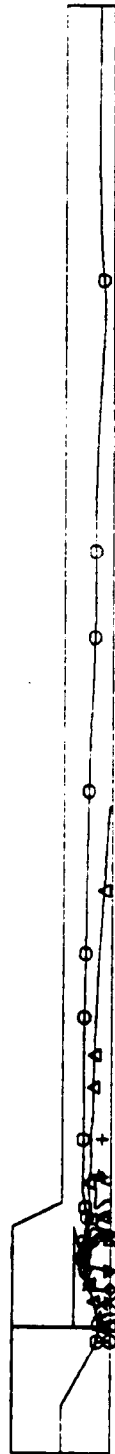


Figure 65. Cell pressure as a function of iteration count for Case 5

○	5.000×10^0
△	1.000×10^0
+	1.500×10^0
×	2.000×10^0
○	2.500×10^0
▽	3.000×10^0
□	3.500×10^0
×	4.000×10^0
◆	4.500×10^0
●	5.000×10^0
■	5.500×10^0
m	6.000×10^0

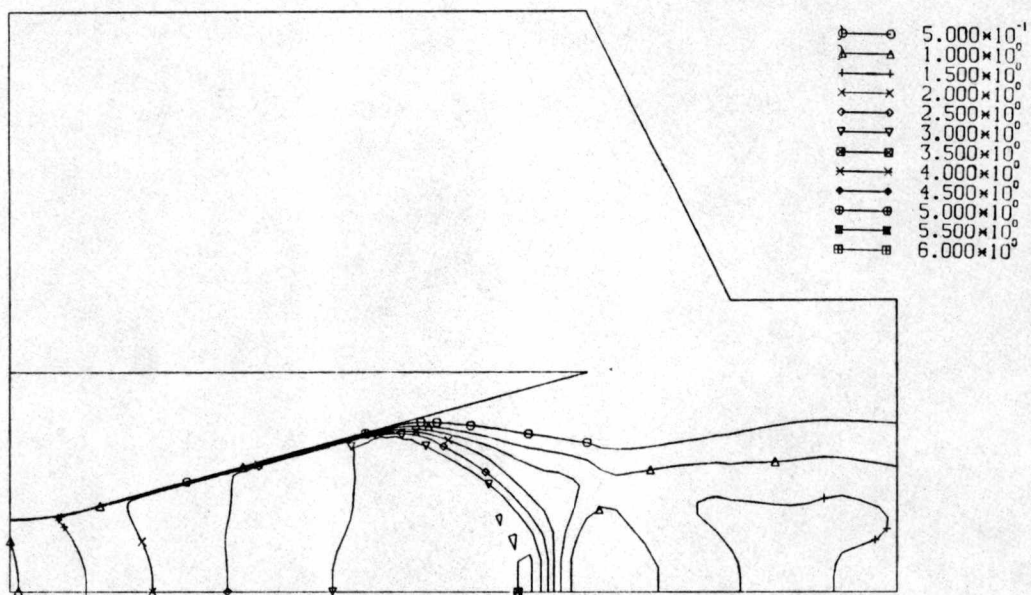


a. Mach number contours

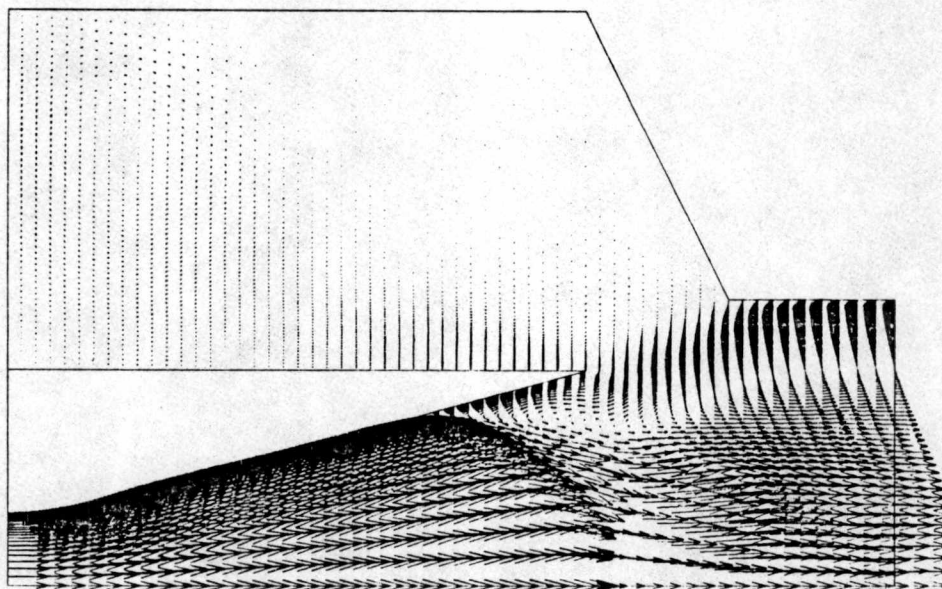


b. Velocity vectors

Figure 66. Flow solution for Case 5



a. Mach number contours



b. Velocity vectors

Figure 67. Nozzle flow solution for Case 5

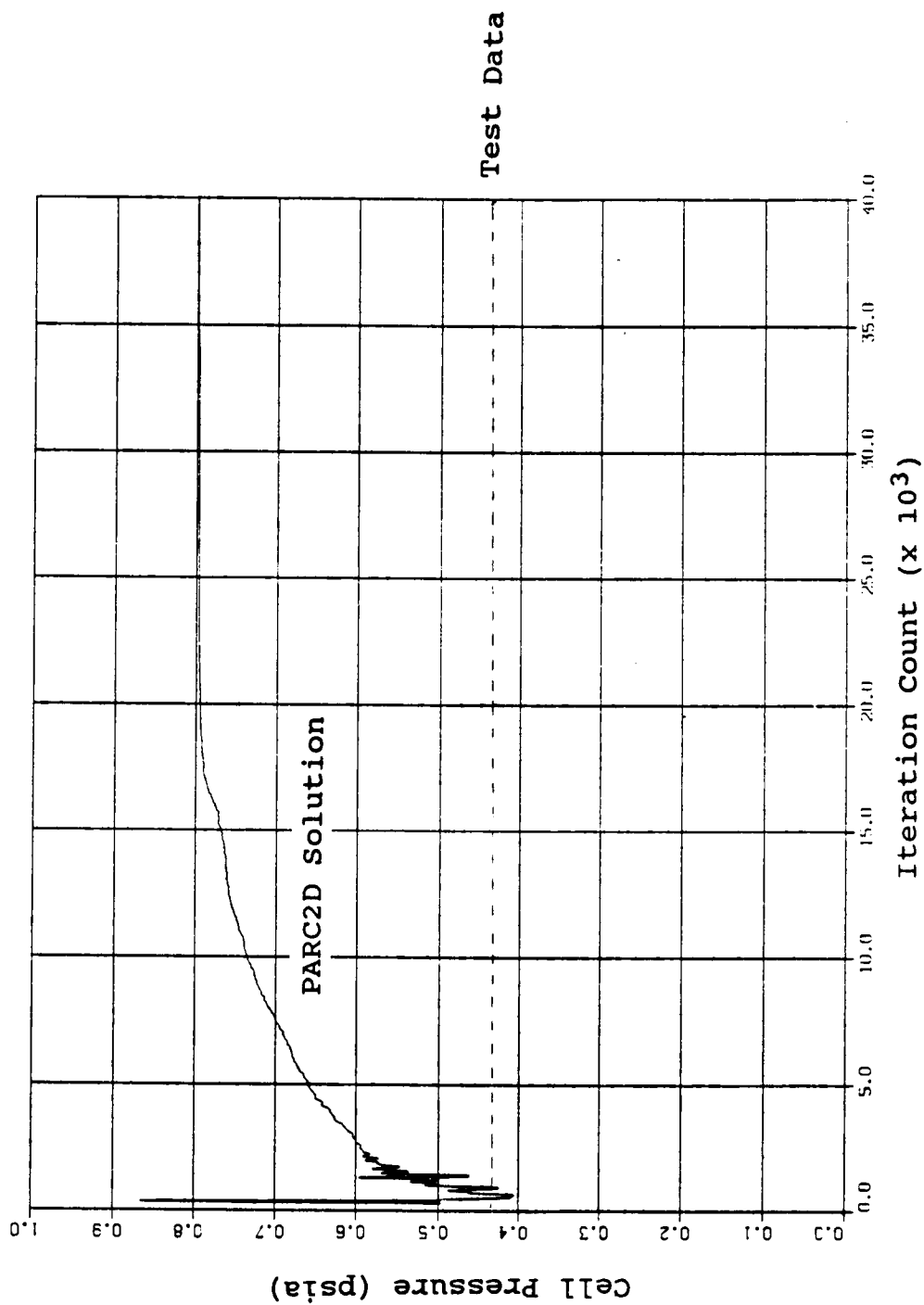
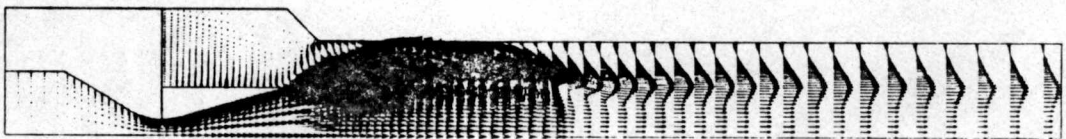


Figure 68. Cell pressure as a function of iteration count for Case 1 with the modified turbulence model

○	5.000×10^{-1}
△	1.000×10^0
+	1.500×10^0
×	2.000×10^0
◇	2.500×10^0
▽	3.000×10^0
■	3.500×10^0
×	4.000×10^0
●	4.500×10^0
⊗	5.000×10^0
■	5.500×10^0
⊗	6.000×10^0



a. Mach number contours



b. Velocity vectors

Figure 69. Flow solution for Case 1 with the modified turbulence model

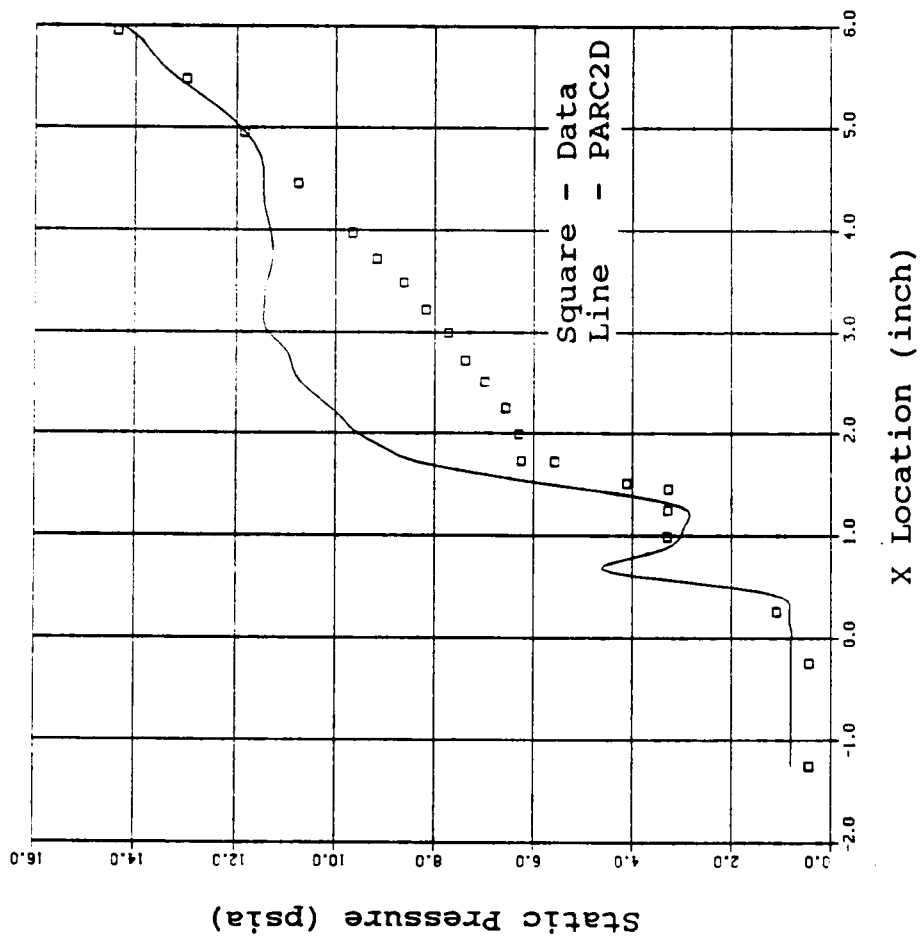


Figure 70. Diffuser wall static pressure comparison for Case 1 with the modified turbulence model

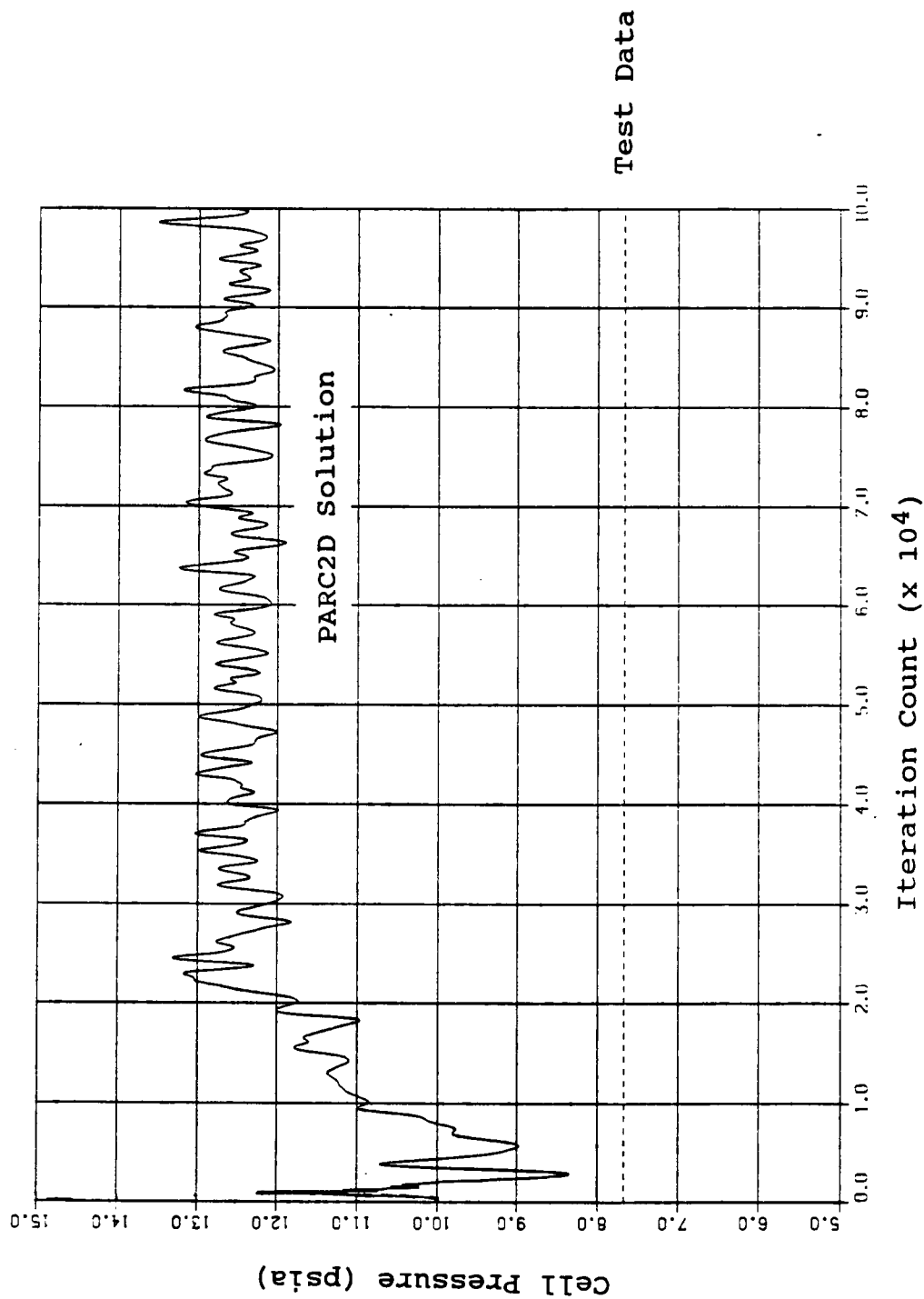


Figure 71. Cell pressure as a function of iteration count for Case 3 with the modified turbulence model

○—○	5.000×10^{-1}
△—△	1.000×10^0
□—□	1.500×10^0
×—×	2.000×10^0
◇—◇	2.500×10^0
▽—▽	3.000×10^0
■—■	3.500×10^0
×—×	4.000×10^0
●—●	4.500×10^0
○—○	5.000×10^0
■—■	5.500×10^0
■—■	6.000×10^0

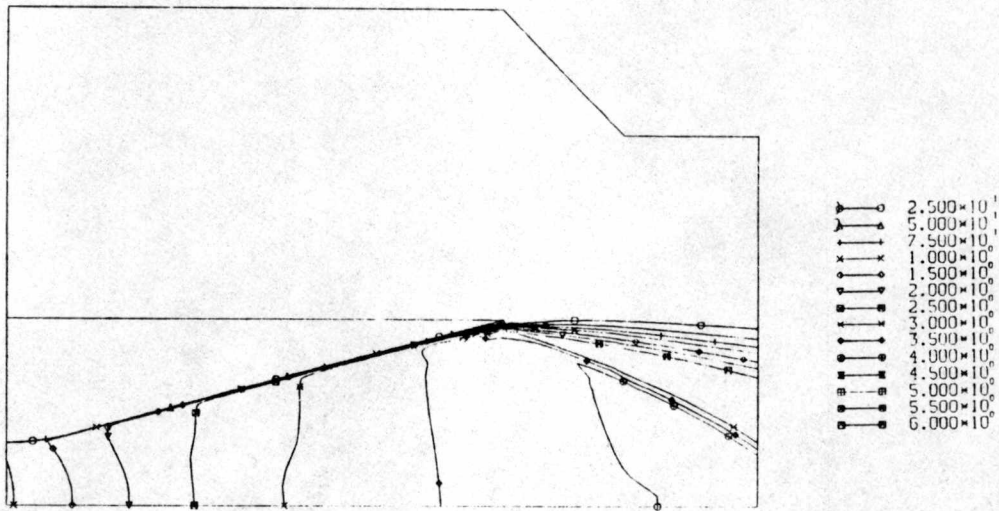


a. Mach number contours

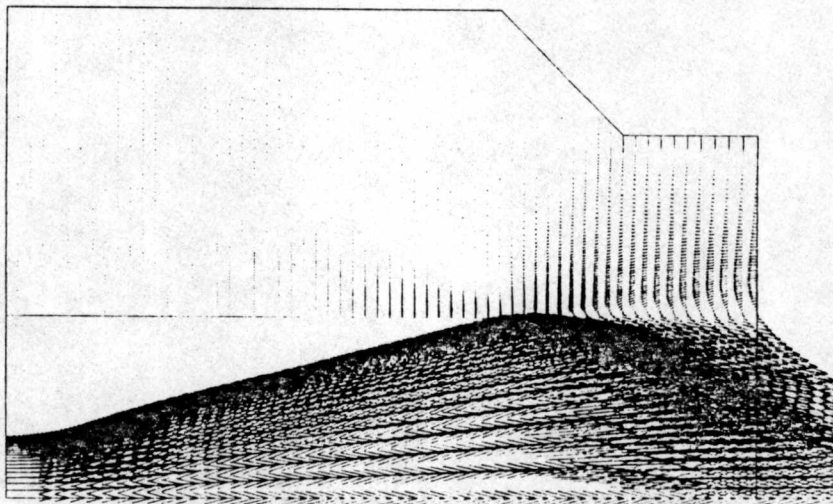


b. Velocity vectors

Figure 72. Flow solution for Case 3 with the modified turbulence model



a. Mach number contours



b. Velocity vectors

Figure 73. Nozzle flow solution for Case 3 with the modified turbulence model

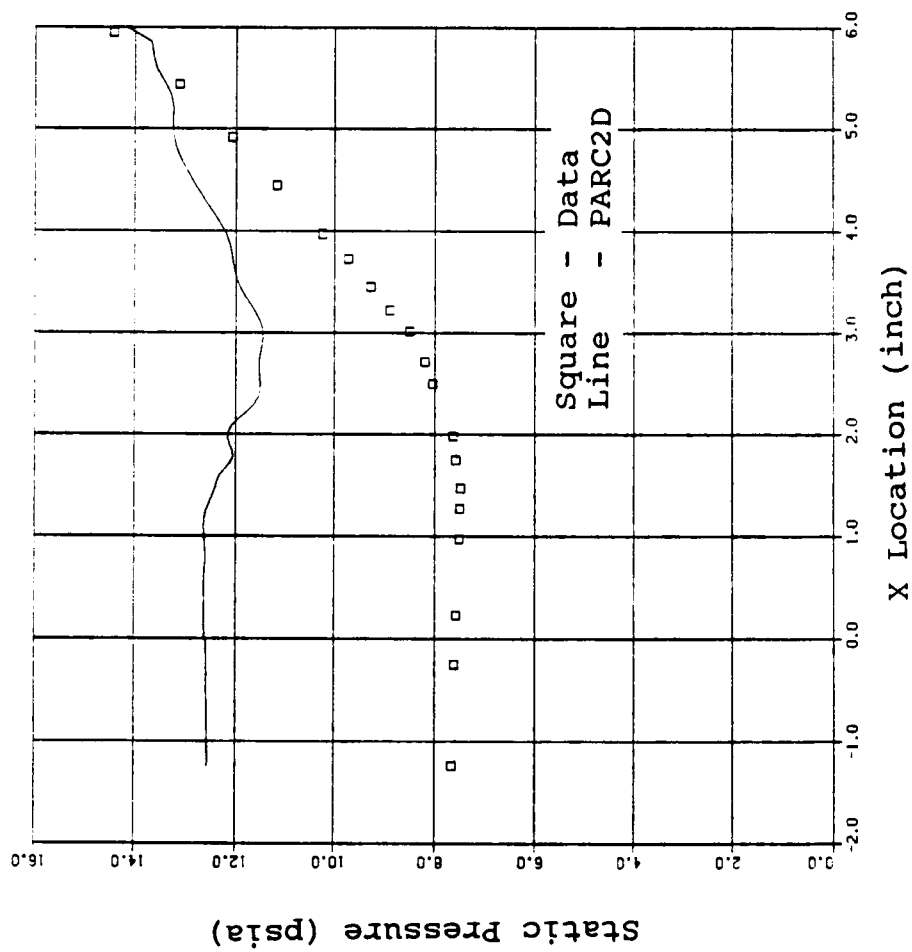


Figure 74. Diffuser wall static pressure comparison for Case 3 with the modified turbulence model

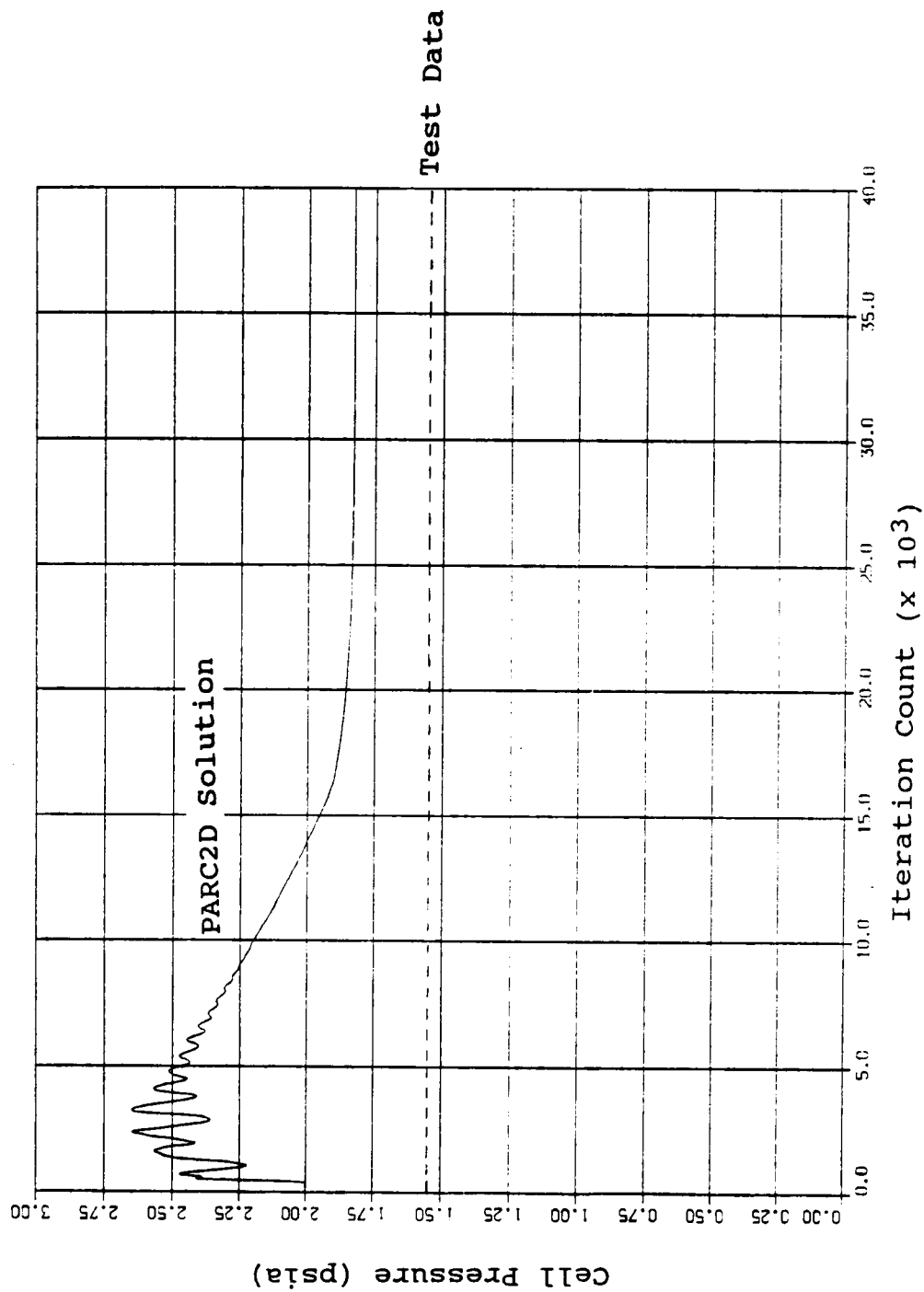


Figure 75. Cell pressure as a function of iteration count for Case 4 with the modified turbulence model

○	5.000×10^0
△	1.000×10^0
+	1.500×10^0
x	2.000×10^0
◇	2.500×10^0
▽	3.000×10^0
■	3.500×10^0
×	4.000×10^0
●	4.500×10^0
⊙	5.000×10^0
■	5.500×10^0
⊞	6.000×10^0



a. Mach number contours



b. Velocity vectors

Figure 76. Flow solution for Case 4 with the modified turbulence model

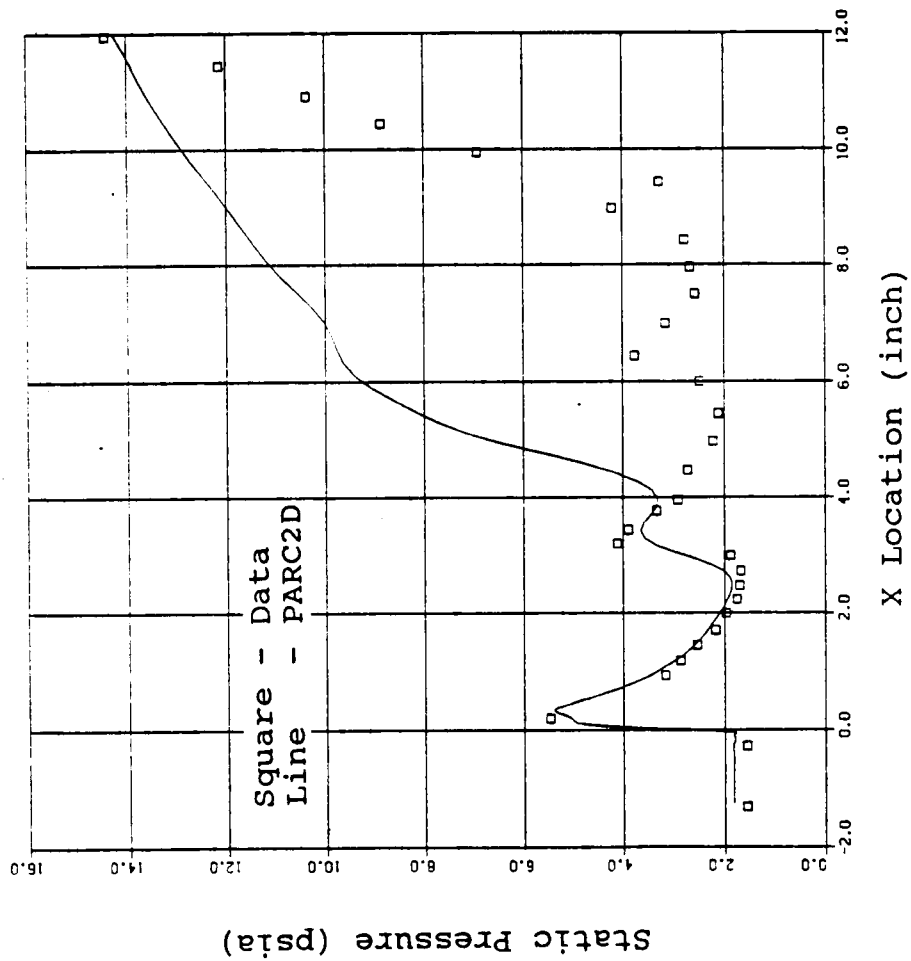


Figure 77. Diffuser wall static pressure comparison for Case 4 with the modified turbulence model

APPENDIX B

PARC2D THEORY

This appendix is presented to describe in a general manner the fundamental equations used by the PARC2D program [30] and to discuss the basic approach taken to solve them. Derivation of the Navier-Stokes equations and the details of the implementation of the algorithms used to solve them will not be discussed here, as they are well documented elsewhere [31, 32, and 33]. The PARC2D program is based on the Navier-Stokes equations expressed in strong conservation law form. It is a flow field simulation program that determines the thermodynamic and kinematic properties of a fluid flow at discrete points in the physical plane based on given boundary conditions. The flow can be treated as either laminar or turbulent, and the viscous terms can be selectively calculated so that a thin-layer simulation can be performed. Inviscid, or Euler, flow fields can also be simulated. The PARC2D thermodynamic relationships are based on the assumptions of an ideal gas with constant ratio of specific heats. A constant Prandtl number is assumed, and the Sutherland viscosity law is used. The functional form for the second coefficient of viscosity is based on the Stokes hypothesis. The governing partial differential equations used

in the PARC2D program are the Reynolds-averaged Navier-Stokes equations in curvilinear coordinates. The equations for continuity, momentum, and energy can be expressed in the following nondimensional conservation law form.

$$\frac{\partial Q}{\partial t} + \frac{\partial F_j}{\partial X_j} = \frac{1}{Re} \frac{\partial G_j}{\partial X_j}$$

The conservation variables vector (Q) is defined as:

$$Q = \begin{bmatrix} \rho \\ \rho u_j \\ E \end{bmatrix}$$

Likewise, the inviscid flux vectors (F_j) are defined as:

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

Finally, the viscous flux vectors (G_j) are represented by:

$$G_j = \begin{bmatrix} 0 \\ \tau_{ij} \\ u_k \tau_{jk} - q_j \end{bmatrix}$$

The viscous stress tensor (τ_{ij}) and the heat flux vector (q_j) take the form:

$$\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial X_j} + \frac{\partial u_j}{\partial X_i} \right) + \lambda \frac{\partial u_k}{\partial X_k} \delta_{ij}$$

$$q_j = - \frac{K}{\beta_r \text{Pr}} \frac{\partial T}{\partial X_j}$$

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

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

For notational convenience, the Einstien summation convention has been made use of.

To apply the above equations for an axisymmetric problem, the following changes are imposed on the two-dimensional equations. First, a source term must be included in the Y-momentum equation:

$$\text{RHS}_{\text{axi}} = \text{RHS}_{2d} + \frac{1}{Y} \left[P - \frac{1}{\text{Re}} \left(2\mu \frac{1}{Y} v + \lambda \nabla_a \cdot u \right) \right]$$

where

$$\nabla_a \cdot u = \frac{1}{Y} v + u_x + v_y$$

Lastly, everywhere the two dimensional divergence $(u_x + u_y)$ appears in the viscous terms, they are replaced by the axisymmetric divergence. Note that it has not been found to be necessary to modify the implicit part of the numerical algorithm.

When turbulence is required, the algebraic turbulence model based on the Thomas formulation [28] of the Baldwin and Lomax model [29] is used. The viscous coefficients are modified as follows:

$$\mu_{\text{total}} = \mu + \mu_T$$

$$\frac{K_{\text{total}}}{Pr} = \frac{K}{Pr} + \frac{\mu_T}{Pr_T}$$

where μ_T is the turbulent viscosity and Pr_T is the turbulent Prandtl number.

The model is made up of two major parts: an unbounded flow model and a bounded flow model. The unbounded flow model takes each coordinate line for each coordinate direction and divides it into sections determined by the location of boundaries and vorticity zeros. Within each section the turbulent viscosity is calculated as

$$\mu_T = Re \rho l V$$

$$V = l \omega$$

$$l = l_0 \frac{[\text{Max}(|u_j|) - \text{Min}(|u_j|)]}{\omega_c}$$

$$|u_j| = \sqrt{u_k u_k}$$

where ω_c is the value of the vorticity at the point where $|u_j|/\omega$ is a maximum, and l_0 is an adjustable constant. The vorticity is defined as usual:

$$\omega = |v_x - u_y|$$

These turbulent viscosity values are smoothed to alleviate sudden changes in viscosity levels.

The bounded flow model is applied near no-slip boundary surfaces and replaces the unbounded flow values with bounded flow values of the turbulent viscosity between the boundary and the point where they match. The formulation is as follows:

$$\mu_T = \text{Re} \rho l V$$

$$l = l_0 d \left(1 - e^{-\frac{d^+}{A^+}} \right)$$

$$d = \sqrt{(x - x_w)^2 + (y - y_w)^2}$$

$$d^+ = \sqrt{\frac{\text{Re} \rho_w \omega d^2}{\mu_w}}$$

$$V = l \omega$$

where l_0 is the Von Karman constant and A^+ is the Van Driest constant; variables subscripted by "w" are evaluated at the boundary.

VITA

Glenn Douglas Garrard was born in Hamilton County, Tennessee, on February 12, 1961. He attended Westwood Elementary and Junior High Schools in Manchester, Tennessee and graduated from Coffee County Central High School in May 1979. He entered Tennessee Technological University the fall of 1979, and in June 1983, he received a Bachelor of Science degree in Mechanical Engineering. Following graduation, he accepted employment with Sverdrup Technology, Inc. at Arnold Engineering Development Center. He began taking graduate courses in September 1983 at the University of Tennessee Space Institute, and graduated with a Master of Science Degree in Mechanical Engineering in May, 1990. Mr. Garrard is an associate member of the American Society of Mechanical Engineers and a member of the American Institute of Aeronautics and Astronautics. He is a member of the First Presbyterian Church in Murfreesboro, Tennessee, and is an Eagle Scout. He is married to the former Angelia Gail Vaughn of Manchester, Tennessee.