

OSURF Project  
#733633  
NASA Grant #  
NAG3-2017

NUMERICAL ANALYSIS  
of  
UNSTEADY FLOW  
in a  
CONVERGING-DIVERGING NOZZLE

submitted by

James N. Scott  
Department of Aerospace Engineering, Applied Mechanics  
& Aviation  
The Ohio State University, Columbus, OH 43210

# Numerical Analysis of Noise Production in a Transonic Converging-Diverging Nozzle

## TABLE OF CONTENTS

|                                                |     |
|------------------------------------------------|-----|
| List of Tables . . . . .                       | xi  |
| List of Figures . . . . .                      | xii |
| Chapters:                                      |     |
| 1. INTRODUCTION . . . . .                      | 1   |
| 2. MOTIVATION AND PREVIOUS WORK . . . . .      | 4   |
| 2.1 Motivation . . . . .                       | 4   |
| 2.2 Previous Work . . . . .                    | 9   |
| 2.2.1 Theory of Jet Noise Production . . . . . | 9   |
| 2.2.2 Analysis of Jet Noise . . . . .          | 12  |
| 3. PROBLEM DESCRIPTION . . . . .               | 17  |
| 3.1 Overview . . . . .                         | 17  |
| 3.2 Nozzle Characteristics . . . . .           | 18  |
| 3.3 Critical Flow Features . . . . .           | 18  |

|       |                                                                  |    |
|-------|------------------------------------------------------------------|----|
| 4.    | SOLUTION METHODOLOGY . . . . .                                   | 21 |
| 4.1   | Computational Domain . . . . .                                   | 21 |
| 4.2   | Computational Grid . . . . .                                     | 23 |
| 4.3   | Governing Equations . . . . .                                    | 25 |
| 4.3.1 | Conservative Forms of Differential Equations . . . . .           | 25 |
| 4.3.2 | Conservation of Mass . . . . .                                   | 25 |
| 4.3.3 | Conservation of Momentum . . . . .                               | 26 |
| 4.3.4 | Conservation of Energy . . . . .                                 | 27 |
| 4.3.5 | Equations of State . . . . .                                     | 28 |
| 4.3.6 | Vector Form of the Conservation Equations . . . . .              | 29 |
| 4.3.7 | Coordinate Transformation . . . . .                              | 30 |
| 4.4   | Numerical Method . . . . .                                       | 32 |
| 4.4.1 | Stability and Accuracy . . . . .                                 | 34 |
| 4.4.2 | Damping . . . . .                                                | 36 |
| 4.5   | Boundary Conditions . . . . .                                    | 37 |
| 4.5.1 | Center Line Boundary Conditions . . . . .                        | 37 |
| 4.5.2 | Far Field Boundary Conditions . . . . .                          | 39 |
| 4.5.3 | Inflow Boundary Conditions . . . . .                             | 40 |
| 4.5.4 | Outflow Boundary Conditions . . . . .                            | 43 |
| 4.5.5 | Nozzle Wall Boundary Conditions . . . . .                        | 46 |
| 4.5.6 | Periodic Boundary Conditions . . . . .                           | 47 |
| 4.6   | Visualization Techniques and Tools . . . . .                     | 47 |
| 4.7   | Unsteady Analysis . . . . .                                      | 50 |
| 4.7.1 | Computation of Unsteady Quantities . . . . .                     | 50 |
| 4.7.2 | Turbulence Calculations . . . . .                                | 53 |
| 4.7.3 | The Convection Speed and Cross Correlation Coefficient . . . . . | 55 |
| 4.7.4 | Scaling Lengths . . . . .                                        | 57 |
| 5.    | NUMERICAL RESULTS . . . . .                                      | 60 |
| 5.1   | General Behavior . . . . .                                       | 60 |
| 5.2   | Flow Visualizations . . . . .                                    | 62 |
| 5.2.1 | Entropy and Jet Spreading Behavior . . . . .                     | 62 |
| 5.2.2 | Shock Locations . . . . .                                        | 68 |
| 5.3   | Turbulence Predictions . . . . .                                 | 75 |
| 5.3.1 | Radial Distribution of Axial Velocity . . . . .                  | 76 |
| 5.3.2 | Radial Distribution of Axial Turbulence Intensity . . . . .      | 84 |
| 5.3.3 | Axial Distribution of Turbulence Intensity . . . . .             | 91 |
| 5.3.4 | Cross Correlation and Convection Speed Predictions . . . . .     | 95 |

|     |                                             |     |
|-----|---------------------------------------------|-----|
| 6.  | CONCLUSIONS AND FUTURE DIRECTIONS . . . . . | 101 |
| 6.1 | Conclusions . . . . .                       | 101 |
| 6.2 | Directions for Future Work . . . . .        | 104 |
|     | Bibliography . . . . .                      | 106 |

## LIST OF TABLES

| Table                                                                | Page |
|----------------------------------------------------------------------|------|
| 3.1 Design Parameters for C-D Nozzle (from [32]) . . . . .           | 19   |
| 4.1 Axial Locations of Radial Velocity Profile Sampling . . . . .    | 51   |
| 4.2 Locations and Spacing for Cross Correlation Sampling . . . . .   | 53   |
| 5.1 Flow Conditions for Simulation Cases . . . . .                   | 61   |
| 5.2 Locations and Angles of Shock Wave and Secondary Compression . . | 75   |
| 5.3 Summary of Momentum Thicknesses Results . . . . .                | 83   |
| 5.4 Summary of $U = 50\%U_j$ Radial Location Results . . . . .       | 84   |
| 5.5 Summary of $U_c/U_j$ Predictions . . . . .                       | 100  |

## LIST OF FIGURES

| Figure                                                                   | Page |
|--------------------------------------------------------------------------|------|
| 2.1 Three Dimensional Modes of Jet Flow . . . . .                        | 8    |
| 2.2 Mixing Noise Propagation (adapted from [28]) . . . . .               | 10   |
| 2.3 Shock Noise Propagation and Screech (adapted from [27]) . . . . .    | 11   |
| 4.1 Computational Domain . . . . .                                       | 22   |
| 4.2 2D Grid Section for Planes K=1 and K=19 . . . . .                    | 23   |
| 4.3 Closeup of 2D Grid Section Near Inflow Boundary . . . . .            | 24   |
| 4.4 Notation for Turbulence and Cross Correlation Calculations . . . . . | 50   |
| 4.5 Locations of Velocity Sampling . . . . .                             | 51   |
| 5.1 Entropy for Run 1 at 43.2 ms. (65000 Iterations) . . . . .           | 64   |
| 5.2 Entropy for Run 2 at 57.8 ms. (85000 Iterations) . . . . .           | 65   |
| 5.3 Entropy for Run 3 at 52.0 ms. (75000 Iterations) . . . . .           | 66   |
| 5.4 Entropy for Run 4 at 58.3 ms. (85000 Iterations) . . . . .           | 67   |
| 5.5 Shock Function for Run 1 at 43.2 ms. (65000 Iterations) . . . . .    | 70   |
| 5.6 Shock Function for Run 2 at 57.8 ms. (85000 Iterations) . . . . .    | 71   |
| 5.7 Shock Function for Run 3 at 52.0 ms. (75000 Iterations) . . . . .    | 73   |

|      |                                                                                                                                      |    |
|------|--------------------------------------------------------------------------------------------------------------------------------------|----|
| 5.8  | Shock Function for Run 4 at 58.3 ms. (85000 Iterations) . . . . .                                                                    | 74 |
| 5.9  | Radial Velocity Profiles from Run 1 . . . . .                                                                                        | 77 |
| 5.10 | Radial Velocity Profiles from Run 2 . . . . .                                                                                        | 78 |
| 5.11 | Streamlines Near Nozzle Separation Region . . . . .                                                                                  | 78 |
| 5.12 | Radial Velocity Profiles from Run 3 . . . . .                                                                                        | 79 |
| 5.13 | Radial Velocity Profiles from Run 4 . . . . .                                                                                        | 80 |
| 5.14 | Radial Velocity Profiles from Preliminary Axisymmetric Calculations<br>by Scott [26] . . . . .                                       | 81 |
| 5.15 | Comparison of Radial Velocity Profiles at $\eta/D = 8.86$ with Experimen-<br>tal Data from Yamamoto, et al. [32] . . . . .           | 82 |
| 5.16 | Radial Turbulence Intensity Profiles from Run 1 . . . . .                                                                            | 85 |
| 5.17 | Radial Turbulence Intensity Profiles from Run 2 . . . . .                                                                            | 86 |
| 5.18 | Radial Turbulence Intensity Profiles from Run 3 . . . . .                                                                            | 87 |
| 5.19 | Radial Turbulence Intensity Profiles from Run 4 . . . . .                                                                            | 88 |
| 5.20 | Radial Turbulence Intensity Profiles from Preliminary Axisymmetric<br>Calculations by Scott [26] . . . . .                           | 89 |
| 5.21 | Comparison of Radial Turbulence Intensity Profiles at $\eta/D = 8.86$ with<br>Experimental Data from Yamamoto, et al. [32] . . . . . | 90 |
| 5.22 | Lip Line Axial Turbulence Intensity Profile from Run 1 . . . . .                                                                     | 92 |
| 5.23 | Lip Line Axial Turbulence Intensity Profile from Run 2 . . . . .                                                                     | 92 |
| 5.24 | Lip Line Axial Turbulence Intensity Profile from Run 3 . . . . .                                                                     | 93 |
| 5.25 | Lip Line Axial Turbulence Intensity Profile from Run 4 . . . . .                                                                     | 93 |

|                                                                                                                             |    |
|-----------------------------------------------------------------------------------------------------------------------------|----|
| 5.26 Comparison of Lip Line Axial Turbulence Intensity Profiles with Experimental Data from Yamamoto, et al. [32] . . . . . | 94 |
| 5.27 Cross Correlation for Run 1 . . . . .                                                                                  | 96 |
| 5.28 Cross Correlation for Run 2 . . . . .                                                                                  | 96 |
| 5.29 Cross Correlation for Run 3 . . . . .                                                                                  | 97 |
| 5.30 Cross Correlation for Run 4 . . . . .                                                                                  | 98 |
| 5.31 Cross Correlation for Low Speed Jet (from [4]) . . . . .                                                               | 99 |

# CHAPTER 1

## INTRODUCTION

As NASA continues development of a High Speed Civil Transport (HSCT), the need for an enhanced understanding of the physics of jet noise grows. The reduction of aircraft noise is a major hurdle in HSCT development, as the federal government has imposed strict guidelines in the allowable noise levels in and around airports. However, the need for reduced engine noise must also be traded off against performance requirements.

The unsteady, turbulent flow in jet engine exhaust is known to be a major source of noise [17, 19, 25]. In an imperfectly expanded supersonic jet flow, shocks will form in the jet core to equilibrate the pressure with that of the surrounding air. Unsteady fluctuations in the location and strength of these shocks generate perturbations which are then convected downstream as pressure waves [12]. Turbulent mixing both within the jet core and in the shear layer between the jet core and the surrounding flow will generate additional pressure waves [28]. In some cases, the instability driving the turbulent mixing and the shock structure interact in a resonant fashion, creating a phenomenon known as screech [19, 27].

Theoretical work with respect to jet noise has produced some understanding of the underlying physics [28, 27]. It is known that jet noise in supersonic flows is comprised of components from turbulent mixing and the shock structure, and theoretical methods have been able to deal with these. However, the simplifying assumptions required to make such a complex problem tractable from a mathematical standpoint also require the analyst to neglect the interaction between important features of the flow physics such as compressibility, three dimensional effects, and the shock cell/instability wave interaction.

Jet noise has been also studied extensively in an experimental context [4, 5, 7, 18, 32]. These studies have revealed much about the structure of the shear layer and the rate at which the jet core mixes with the surrounding flow. While experimental techniques have contributed and will continue to contribute much to the understanding of jet noise, the expense of building experiments and the difficulties associated with real-time data acquisition are limiting factors on what can be studied experimentally.

Several researchers have also approached the jet noise problem using numerical methods [1, 3, 11, 12, 17, 19, 22, 23, 25]. The development of a numerical simulation is potentially much less costly than the configuration and maintenance of experimental facilities, and once a simulation technology is developed it can easily be applied to a range of operating conditions. However, the complexity of the physics and the massive amount of computer resources required for an accurate simulation have caused previous investigations to restrict themselves to two dimensional planar or axisymmetric configurations for simulations of jet noise. Many studies have further eased the computational requirements of their simulations by simulating the time

average flow rather than the full unsteady flow, using a turbulence model to account for the unsteady behavior neglected by the steady state assumption.

This study investigates the physics of jet noise by the numerical solution of the three dimensional unsteady Navier-Stokes equations for a low-supersonic jet (Mach numbers on the order of 1.4) from a converging-diverging (C-D) nozzle and mixing with a subsonic surrounding flow. The numerical method used for the solution of these equations is the second order accurate MacCormack explicit predictor-corrector scheme. The numerical results are compared to the experimental results of Yamamoto for the same C-D nozzle geometry and similar operating conditions. The goal of the study is to use numerical simulation techniques to resolve accurately physical processes which contribute to jet noise, and to assess whether performing these simulations in a three dimensional geometry is worth the additional cost in computing resources as compared to a comparable simulation on a two dimensional planar or axisymmetric geometry. Scientific visualization is used to observe the spreading characteristics of the jet, as well as to examine the structure of the shocks within the jet core. The time average velocity and turbulence intensity level at various locations are computed and compared with experimental results. A two point space-time cross correlation is used to compute the mean convection speed through the shear layer, and these predictions are compared with results from theory.

## CHAPTER 2

### MOTIVATION AND PREVIOUS WORK

This chapter outlines the motivation for investigating transonic jet flow using a numerical simulation approach. Advantages and disadvantages of using a numerical approach are described. Furthermore, an overview of previous work on jet noise using theoretical, experimental, and numerical methods is given.

#### 2.1 Motivation

The primary motivation for this study is the need for a more detailed understanding of the physics underlying the production and propagation of jet noise. This understanding is necessary for the development of advanced propulsion systems such as those proposed for the High Speed Civil Transport (HSCT). Not only is jet noise a major nuisance for those who must live or work nearby, but it can also contribute to vibration leading to structural fatigue in aircraft structures.

Many researchers have examined the production of jet noise from the theoretical, experimental, and numerical points of view. In these studies, it has been observed that jet noise production is driven primarily by unsteady flow behavior which includes the formation and propagation of vortices, especially in the shear layer between the core jet flow and the surrounding flow. These studies have also shown that the speed

at which disturbances propagate through the shear layer is a function of the flow and sound speeds both within the jet core and in the surrounding flow.

There are several benefits to investigating jet noise production using a numerical approach. By far the most important benefit to investigating jet noise numerically is that flow properties can be computed at any location within the domain of the computation without influencing the characteristics of the flow. In experimental studies, some form of probe is typically used to measure pressure, temperature, air speed, and so forth in the flow. While these probes are usually quite small, their very presence can influence the behavior of a complex flow in ways which are difficult to predict *a priori*. Furthermore, most commonly available probes measure one or possibly two flow quantities. In a numerical simulation, computing predictions at a location does not require any intrusion into the flow which can modify the flow's behavior, and the types of predictions which can be computed are limited only by the simulation method and the researcher's imagination.

In supersonic, turbulent flows such as those encountered in jet noise research, there is an additional benefit to a numerical simulation over an experimental setting. At supersonic speeds, it is extremely difficult to fabricate probes which can react at a frequency high enough to resolve the smaller time scales caused by the higher speeds. Because of this, certain promising analysis methods have only been applied to low speed jets, such as the two point space-time correlation used by Chu [4] for predicting convection speed. In an unsteady numerical simulation, the only limiting factors on frequency response are the stability requirements of the numerical method used and the range of length scales in the computational grid.

While there have been numerous numerical investigations of jet noise before this study, almost all of them have been for either two dimensional or strictly axisymmetric configurations. There are, however, several advantages to doing a full three dimensional simulation over the simplified two dimensional and axisymmetric cases. One advantage of simulating in three dimensions is the influence of a numerical phenomenon known as three-dimensional pressure relief. In numerical simulations of flow constrained to be in a single plane, directional momentum can only be in one of two orthogonal directions. Any other momentum then becomes randomly directed momentum, which is represented by pressure. In a three dimensional simulation of flow, directional momentum can be in three orthogonal directions, and thus there is less momentum to be expressed in terms of pressure. This means that two dimensional and axisymmetric simulations consistently over-predict pressure relative to a three dimensional simulation of the same geometry, and the decrease in pressure seen in three dimensional simulations is referred to as three dimensional pressure relief.

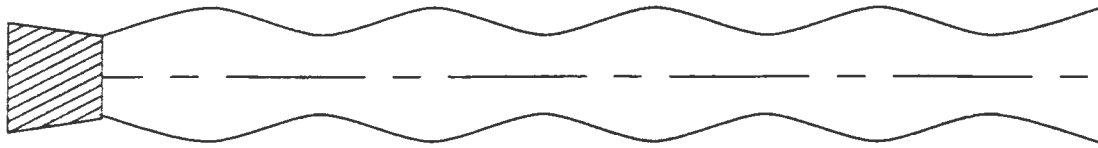
A second reason to simulate flow numerically in three dimensions rather than in two is the influence of additional dissipation inherent in the three dimensional flow equations. In two dimensional viscous flow, the main sources of dissipation are the viscous stress terms  $\tau_{xx}$ ,  $\tau_{xy}$ , and  $\tau_{yy}$  (shown in Equations 4.5, 4.6, and 4.8). In three dimensional viscous flow, there are an additional three viscous stress terms:  $\tau_{xz}$ ,  $\tau_{yz}$ , and  $\tau_{zz}$  (shown in Equations 4.7, 4.9, and 4.10 respectively). The effect of this additional dissipation is to “smear out” sharply discontinuous phenomena such as shock waves. This, in combination with the three dimensional pressure relief described above, means that the shock structures predicted by three dimensional simulations

can be often quite different than those predicted by two dimensional or axisymmetric simulations.

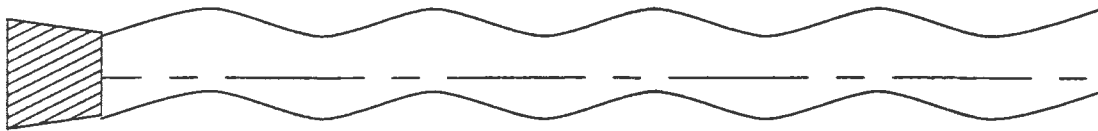
A third advantage of numerical simulations in three dimensions is the influence of three dimensional modes which are expressly denied in planar simulations. For cylindrical jets such as those of interest to study, there are three possible three dimensional modes. The first mode is the varicose or pulsating mode, in which the width of the jet core expands and contracts at a regular frequency. The second mode is the sinuous or flapping mode, where the jet core flaps up and down or side to side. The third mode is the helical or swirling mode, where the jet core spins about the downstream axis. Diagrams of these modes are shown in Figure 2.1

There are, of course, limitations to numerical approaches such as used in this study. First and most telling, it is extremely expensive in terms of computation time. The three dimensional grid used for this study has approximately 325,000 grid nodes, which is large although not outrageously so. However, the time required to compute 5,000 iterations on one processor of a 200 megaflop dual processor computer is roughly 20 hours, meaning that each iteration requires on the order of a *billion* floating point operations. Calculations of this size virtually require the use a computer capable of at least 100 megaflops; less capable machines would make the calculation unreasonably long, as getting usable unsteady results requires at least 35,000 iterations and preferably 75,000 iterations or more.

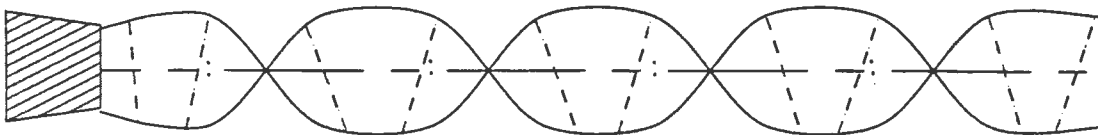
A second limitation on numerical approaches is one of resolution of spatial and temporal scales. At the low end, a simulation is only accurate down to the length scales of the discretization grid used; any phenomena occurring on scales smaller than that of the grid, such as extremely small scale turbulence, will not be resolved



a. Varicose (pulsating) mode



b. Sinuous (flapping) mode



c. Helical (swirling) mode

Figure 2.1: Three Dimensional Modes of Jet Flow

accurately. At the other extreme, a unsteady simulation which is not run for a long enough duration will not resolve unsteady phenomena occurring at extremely low frequencies.

## **2.2 Previous Work**

This section summarizes the current literature regarding unsteady jet flow and its contribution to jet noise. A description of the physics underlying jet noise production is included, as well as a discussion of both experimental and numerical investigations of jet noise. The visualization of unsteady flow results and the numerical prediction of far-field noise levels, two topics peculiar to numerical investigations of jet noise, are also discussed.

### **2.2.1 Theory of Jet Noise Production**

Two considerations in the analysis of jet noise are mixing noise and shock noise [4, 19]. While mixing and shock noise are convected in the same way, they are produced by different physical processes, and this has an important impact on the frequency content of the noise produced. As shown in Figure 2.2, mixing noise is caused by the mixing between the initially uniform jet core and the generally much slower exterior flow. Mixing noise is driven by processes which are inherently viscous and turbulent in nature. Mixing noise has two components, shear mixing noise and self mixing noise [4]. Shear mixing noise is caused by mixing in the shear layer between the jet core and the exterior flow, while self noise is caused by mixing within the jet core. Although self and shear noise are mathematically and physically different, they are virtually indistinguishable experimentally. Mixing noise in general is a broadband effect, in that it can be seen all across the acoustic spectrum. A study by Tam and Burton

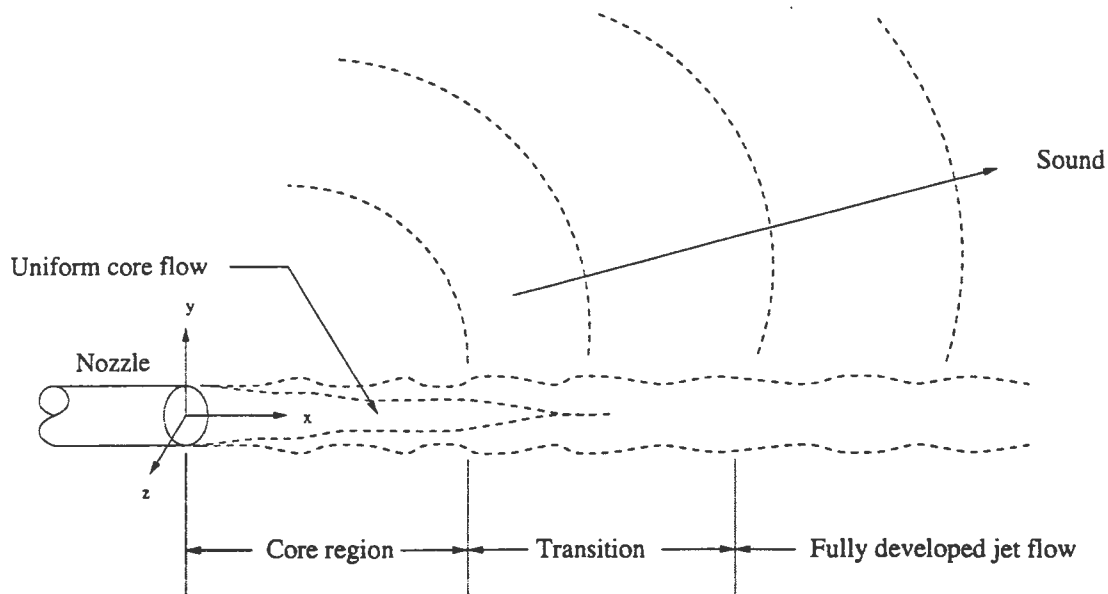


Figure 2.2: Mixing Noise Propagation (adapted from [28])

[28] showed that, for an axisymmetric jet, the spatial evolution of the instability waves which cause mixing noise can be expressed in terms of a matched asymptotic expansion. Furthermore, they found that axisymmetry allowed them to decompose the instability waves into azimuthal modes using a Fourier series expansion.

Shock noise is substantially different from mixing noise, in that it is an effect of both compressibility and viscosity. Shock noise is caused by pressure fluctuations generated by the interaction of the jet shock cell and viscous instability waves. As instability waves move downstream, they cause the shock cell to readjust, and the resulting fluctuation in pressure propagates downstream. In extreme situations, shock noise can lead to a powerful high frequency acoustic phenomenon known as screech, shown in Figure 2.3. In screech, the pressure waves generated by the shock

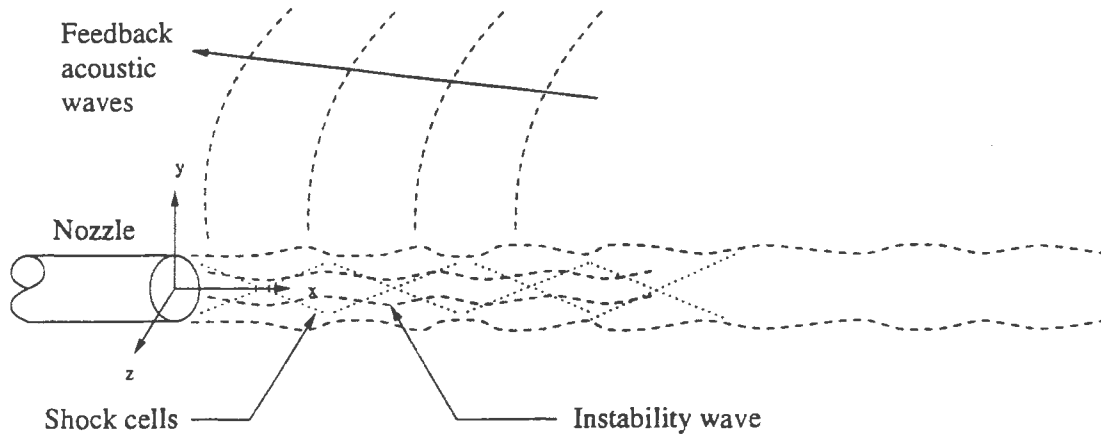


Figure 2.3: Shock Noise Propagation and Screech (adapted from [27])

cell/instability interaction travel back upstream in sufficient strength to cause further instability waves which in turn interact with the shock cell structure to generate more acoustic waves. If the pressure waves interfere constructively with the instability waves, a feedback loop is created which cause the noise levels at certain discrete frequencies to rise to levels which can cause acoustic excitation of vibrations leading to structural fatigue in the aircraft or nearby structures. An investigation by Tam, Ahuja, and Jones [27] showed that screech frequencies are controlled by the character of the feedback pressure waves, which in turn are controlled by the characteristics of the instability waves driving the system. Furthermore, this study showed that super resonance, an extreme form of screech involving extremely large pressure variations, is caused by the instability wave most amplified by the screech feedback loop.

## 2.2.2 Analysis of Jet Noise

### Experimental Work in Jet Noise

Most of the early work in the analysis of jet noise is experimental in nature, particularly before the advent of digital computers. One of the most important early studies of jet noise was that of Chu [4], who examined the influence of turbulence on the production of noise in a low speed jet. Chu's study reformulated the Lighthill acoustic analogy equations [15, 16] in terms of a frequency spectrum using a Fourier series. This showed that the convection of disturbances in a flow is a combination of Doppler shift and amplification of moving eddies. Experimentally, a two point space-time correlation of velocity and its square were used to predict the convection speed, the speed at which a disturbance is convected downstream by the flow. Estimates of the intensity and frequency spectra of the self and shear noise levels generated by turbulence in the jet core were also made.

Another experimental investigation of jet noise was that of Yamamoto, et al. [32], who experimentally investigated the noise characteristics of several nozzles including the converging-diverging (C-D) nozzle examined in this study as well as a simple converging nozzle and several annular plug nozzles. These nozzles were tested at both stationary and simulated flight operating conditions in an anechoic chamber. In addition to acoustic measurements using an array of microphones, the jet plume was examined using a shadowgraph photography system and a laser velocimeter. A theoretical method for estimating shock cell noise in the vicinity of plug nozzle also was developed. Yamamoto's study found that the cylindrical C-D nozzle had the lowest noise characteristics of the nozzles examined, showing a reduction of 11 dB

in perceived noise level at an angle of  $50^\circ$  from the jet axis under simulated flight conditions from that observed in the converging nozzle used as the baseline case.

Numerous other experimental studies of jet noise have been performed. Chuech, Lai and Faeth [5] examined effects of compressibility on the mixing behavior of a nearly sonic jet both experimentally and numerically, using a parabolized Navier-Stokes scheme with a  $k$ - $\epsilon$  turbulence model for the numerical portion of their investigation. Their study confirmed the earlier results of Papamoschou and Roshko [18] that compressibility effects reduce the jet mixing rate for high convective Mach numbers. Furthermore, their study concluded that the numerical turbulence model accurately captured the structure of the jet only in the constant pressure portion of the jet, away from the shock structure. In their experimental study of a supersonic jet exhausting into still air, Fourquette, Mungal, and Dibble [7] provided further confirmation of the results of Papamoschou and Roshko.

### **Numerical Simulation of Jet Noise**

The computational simulation of jet noise is currently a topic of active and aggressive research. The flows which create jet noise have several features of interest to computational fluid dynamics research, including shocks, turbulence, and unsteady behavior. As a result, flows with features of jet noise have been treated with a wide ranges of approaches.

Scott's investigations [22, 23, 25] have centered about the simulation of rectangular and cylindrical jet flows using a MacCormack explicit solver for the unsteady Navier-Stokes equations. In cylindrical jet flows [22, 23], his predictions show excellent agreement with experimental results for shock structure, vorticity, turbulence intensity, and velocity. These studies found that the agreement between numerical

predictions and experimental results tended to improve with longer simulation times and grid refinement. Scott has also computed numerical simulations of the cylindrical jet configuration originally investigated experimentally by Chu [25]. These computations included the calculation of the same two point space-time cross correlation used by Chu to predict the convection speed. The general trends of these predictions agreed with experimental results, and this agreement improved as the length of the simulation increased.

Rozmajzl [19] investigated the flow mechanisms underlying screech in a rectangular jet, also using a MacCormack explicit solver for the unsteady Navier-Stokes equations. In this study, Rozmajzl was able to capture numerically the pressure waves propagating upstream which drives the screech cycle. It also was found that the numerical results of Rozmajzl's study agreed closely with the theoretical predictions made by Tam [27].

Bayliss and Maestrello [3] used a MacCormack explicit solver in their investigation of non-axisymmetric disturbances in subsonic jets. However, their study dealt only with inviscid effects, using the Euler equations rather than the Navier-Stokes equations as the governing equations. Bayliss and Maestrello's study considered primarily flapping modes in cylindrical subsonic jets. They found that the long time (i.e. low frequency) behavior of the jet is dominated by the vortex structure created by the wake of the nozzle lip. This study also found that high frequency acoustic noise tends to be radiated perpendicular to the jet axis, while low frequency noise tends to be radiated at moderate oblique angles to the jet axis.

Al Qadi [1] also used a MacCormack explicit solver in his study of the influence of nozzle lip geometry on mixing. Al Qadi's approach used boundary conditions based

on a rigorous characteristic variable formulation. Al Qadi found that the variation of the nozzle lip geometry can have a substantial effect on the mixing rate of jet as well as the strength and direction of pressure waves generated by mixing and the shock structure within the jet core.

In contrast to the methods described above, Khavaran, Kresja, and Kim [11, 12] used a predominantly steady-state approach in their simulation of the converging-diverging nozzle investigated experimentally by Yamamoto. This simulation used a Beam-Warming alternating direction implicit (ADI) solver for the steady Reynolds-averaged Navier-Stokes equations, with a  $k-\epsilon$  turbulence model. The steady state results were then used to compute the frequency spectrum of the far field acoustic field behavior using the formulation of Mani and Balsa. The results of this method produced usable agreement with Yamamoto's experimental results for sound pressure level, particularly for angles between  $20^\circ$  and  $60^\circ$ . Furthermore, grid refinement was found to improve agreement with aerodynamic data, but it did not seem to affect the mixing noise frequency spectra predicted.

Mankbadi, Hayer, and Povinelli [17] used yet another numerical approach in their simulation of a supersonic cylindrical jet. Their numerical simulation used large eddy simulation (LES) techniques rather than direct solution of the Navier-Stokes equations, allowing them to capture much longer length scales than is practical with a direct Navier-Stokes solution. However, this also requires that turbulent stresses on length scales smaller than that of the grid must be modeled using Smagorinsky's model, meaning that the accuracy of the solution is in large part contingent on the accuracy of the modeling used. Mankbadi, Hayer, and Povinelli acknowledged this

and cautioned that their numerical results could be compared with experiment only in qualitative terms; their study was intended only as a proof of concept.

Colonus, Lele, and Moin [6] examined the important topic of non-reflective boundary conditions for Navier-Stokes solutions. Their formulations of subsonic inflow and outflow boundary conditions were cast in terms of characteristic variables rather than primitive or thermodynamic variables. They found that the combination of characteristic boundary conditions and extreme grid stretching was able to reduce the reflection of vortices from an outflow boundary by up to three orders of magnitude. It also was found that this combination of boundary conditions and grid stretching was sufficient to avoid numerical self-excitation (i.e. non-physical screech).

## CHAPTER 3

### PROBLEM DESCRIPTION

This chapter describes the particulars of the problem investigated in this study. The nozzle to be simulated is described, including the characteristics which make it a candidate for unsteady investigation. Quantities important to describing the noise characteristics of the nozzle are also described.

#### 3.1 Overview

As described in Section 2.2.1, transonic jet noise is a broadband acoustic phenomenon driven by both viscous and compressibility effects in the interaction between a supersonic core flow and a surrounding subsonic flow. Acoustic disturbances are created by turbulent mixing both within the jet core and in the shear layer between the jet core and the surrounding flow. More acoustic disturbances are also produced by the interaction of the jet shock cell structure with the shear layer. While much has been learned from theoretical, experimental, and numerical studies of this type of flow, a more refined understanding of the physics underlying jet noise is required to reduce jet engine noise. In particular, the evolution and behavior of disturbances propagating through the shear layer needs more investigation, especially as it relates to jet spreading.

The goal of this study is to investigate aspects of jet noise by solving the three dimensional time-dependent Navier-Stokes equations for a slightly under-expanded, supersonic, cylindrical jet exiting from a converging-diverging nozzle and mixing with a surrounding subsonic flow. Accurate resolution of flow features which influence the production of jet noise is sought, as well as a better understanding of the mixing process. Numerical predictions acquired from the simulations are compared with the experimental results of Yamamoto [32] as well as predictions from the theory of Papamoschou and Roshko [18] and the numerical calculations of Khavaran, Kresja, and Kim [11, 12]. An assessment is made of the applicability of the method employed versus those used in two dimensional and axisymmetric numerical simulations of the same class of flows.

## 3.2 Nozzle Characteristics

The nozzle examined in this study is the converging-diverging nozzle examined experimentally by Yamamoto [32] and numerically by Khavaran, Kresja, and Kim [11, 12]. The design parameters for this nozzle are shown in Table 3.1. This nozzle has been used primarily as a research nozzle; to the author's knowledge, it has not been used on an operational aircraft. However, its operational conditions are representative of those needed for applications in a High Speed Civil Transport (HSCT) aircraft configuration.

## 3.3 Critical Flow Features

The accurate resolution of certain unsteady features in the jet flow is critical to understanding how jet noise is produced. The most influential of these features is

|                                                 |           |
|-------------------------------------------------|-----------|
| Throat Diameter                                 | 5.1 in.   |
| Exit Diameter                                   | 5.395 in. |
| Distance from Throat to Exit                    | 5.525 in. |
| Exit Velocity                                   | 2409 fps. |
| Ambient Velocity                                | 400 fps.  |
| Pressure Ratio $\left(\frac{p_1^0}{p_e}\right)$ | 3.121     |
| Stagnation Temperature                          | 1716° R   |

Table 3.1: Design Parameters for C-D Nozzle (from [32])

the behavior of the shear layer. The rate at which the shear layer grows in thickness determines how quickly the jet core becomes fully mixed with the surrounding flow. The shear layer is studied by examining time averages of the turbulence intensity and streamwise velocity distributions in both the radial and axial directions at several locations within the flow.

The convection speed, the speed at which disturbances propagate downstream in the shear layer, is also of particular interest to this investigation. This can be estimated by a theoretical prediction, although this prediction is known to underestimate the observed convection speed if compressibility cannot be neglected (as is the case when the jet core is supersonic). The convection speed can also be computed directly using a two point space-time cross correlation.

Another important feature of the jet flow is the strength and orientation of shock waves within the jet core. These influence the shock noise generated by the jet as well as the possible development of a screech cycle. The identification of shock structures within the jet core is handled using scientific visualization techniques; the high gradients in density, pressure, and Mach number created by shocks make their

presence and location easily discernible to the observer. Images generated using these techniques can be readily compared to schlieren and shadowgraph photographs taken in an experimental setting.

## CHAPTER 4

### SOLUTION METHODOLOGY

This chapter describes the theory and methodology used in the unsteady numerical simulation of three dimensional flow through the converging-diverging nozzle described in Chapter 3 and into a large chamber of much slower moving air. The numerical methods used in the simulation process are discussed, as are methods for computing and analyzing predictions from the simulation. Techniques for visualizing behavior of the flow domain as a whole are also outlined.

#### 4.1 Computational Domain

The computational domain used in this study is a circular cylinder approximately twelve nozzle exit diameters in diameter and thirty-two exit diameters in length. The axis of the cylinder runs along the streamwise axis of the nozzle, and one end of the cylinder is coplanar with the inflow plane of the nozzle as shown in Figure 4.1. This configuration allows for a relatively simple treatment of boundary conditions, as described in Section 4.5.

The conventional right-handed Cartesian coordinate frame is used within this domain. The X axis is aligned with the streamwise axis of the nozzle. The Y and Z axes are directed along radial directions at right angles, with the Y axis in the

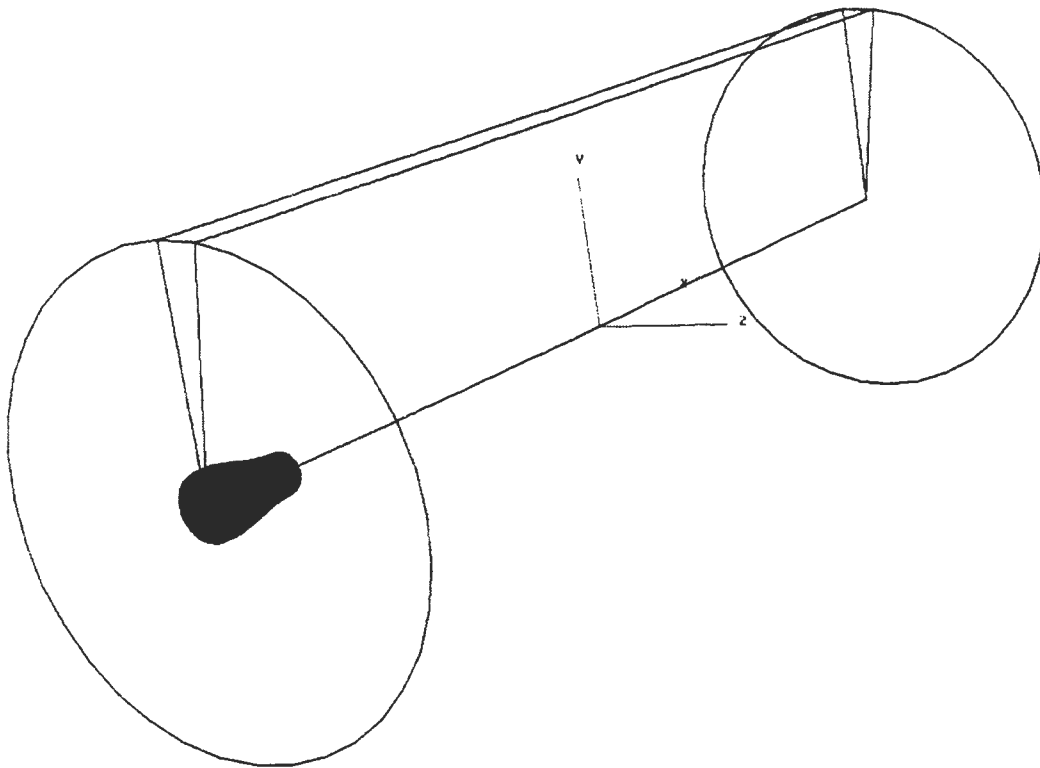


Figure 4.1: Computational Domain

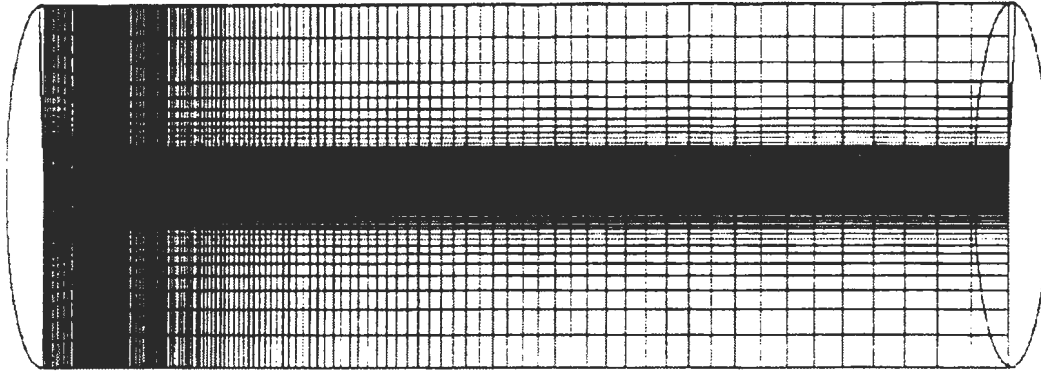


Figure 4.2: 2D Grid Section for Planes  $K=1$  and  $K=19$

$\theta=0^\circ$  half-plane and the  $Z$  axis in the  $\theta=90^\circ$  half-plane. The origin of this coordinate system is located at the center of the inflow plane of the nozzle.

## 4.2 Computational Grid

The grid used to discretize the domain described above is a three dimensional (3D) version of the  $141 \times 61$  grid used by Khavaran, Kresja, and Kim [11]. To generate this 3D grid, the original axisymmetric grid used by Khavaran, Kresja, and Kim was swept about the axis of symmetry in intervals of  $10^\circ$ . Additionally, two overlapping planes are incorporated for periodic boundary conditions as described in Section 4.5.6. The resulting grid is  $141 \times 61 \times 38$ , with a total of 326,838 nodes.

The referencing of a particular point in the grid space is relatively simple. The first index,  $I$ , refers to a location in the axial direction ranging from 1 to 141;  $I=1$  refers to the inflow plane,  $I=50$  to the nozzle exit plane, and  $I=141$  to the outflow plane. The second index,  $J$ , refers to a location in the radial direction ranging from 1 to 61;  $J=1$  refers to the centerline,  $J=25$  to the inner surface of the nozzle wall,  $J=26$  to the

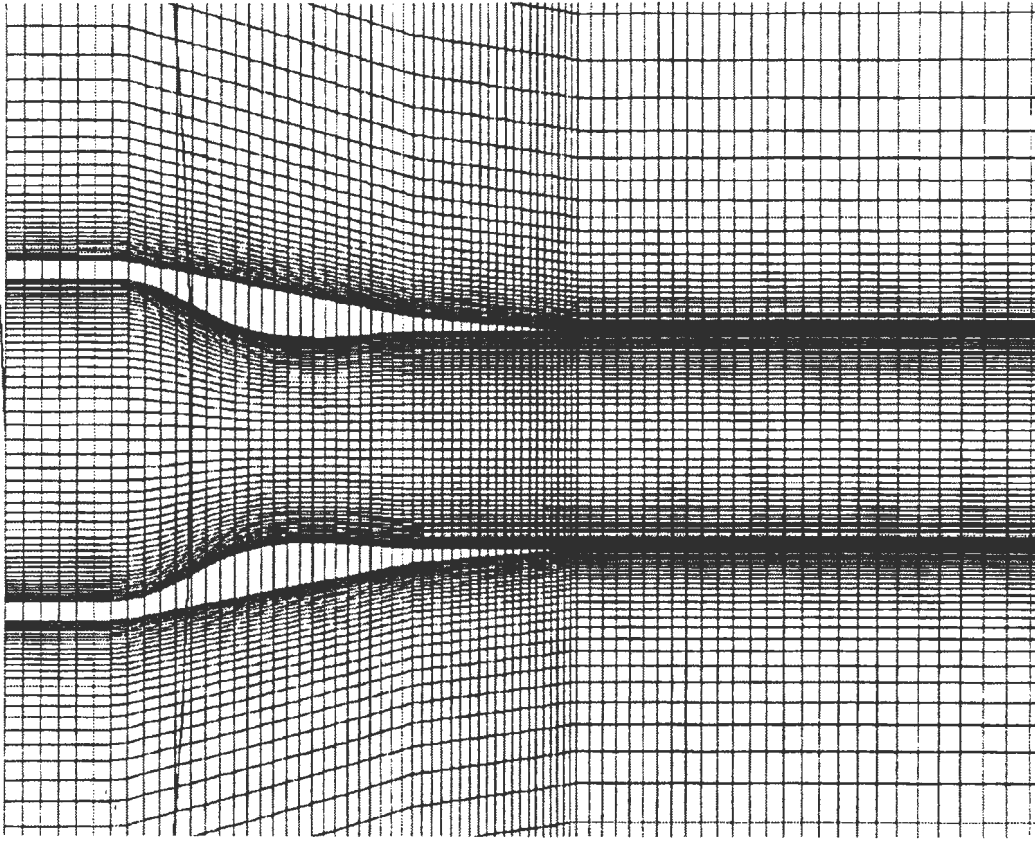


Figure 4.3: Closeup of 2D Grid Section Near Inflow Boundary

outer surface of the nozzle wall, and  $J=61$  to the outer radial extent of the domain. Finally, the third index,  $K$ , refers to an angular location  $\theta$  about the streamwise axis;  $K=1$  refers to the  $\theta=0$  half-plane, which overlaps in physical space with  $K=37$ . Similarly,  $K=2$  and  $K=38$  overlap in physical space for the aforementioned reason of the periodicity condition. The grids for planes  $K=1$  and  $K=19$ , in total as well as near the nozzle, are shown in Figures 4.2 and 4.3.

## 4.3 Governing Equations

The equations of motion used in the simulations for this study are the 3D time-dependent Navier-Stokes equations. The Navier-Stokes equations express the conservation of mass and energy as well as momentum along each of the three Cartesian axes. This results in a total of five interdependent governing equations, which are presented below. Complete derivations of these equations are shown by Anderson, Tannehill, and Pletcher[2] as well as Kuethe and Chow [13].

### 4.3.1 Conservative Forms of Differential Equations

A differential equation is said to be in conservative form if the coefficients of all derivative terms in the equation are either constant or do not appear in any of the other derivatives in the equation [2, 9]. Equations in conservative form have the divergence of a physical quantity identifiable in the equation. As the governing equations used in this study, the Navier-Stokes equations, express the conservation of several physical quantities, it is important that they be solved in conservative form so that the conservation properties of the differential equations hold.

### 4.3.2 Conservation of Mass

The continuity equation expresses conservation of mass in three dimensions. It states a relationship between the time rate of change of mass (usually expressed as a volume integral of the density  $\rho$ ) in a control volume and the rate at which mass flows into or out of the bounding surface of the volume. The conservative form of the continuity equation can be expressed in the differential form given by

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (4.1)$$

### 4.3.3 Conservation of Momentum

Conservation of momentum must be enforced along all three Cartesian axes, resulting in three component momentum equations. These equations describe the relationship between the time rate of change in momentum along a given direction and the forces applied in that direction. The conservative forms of the momentum equations are

X momentum:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2 + p - \tau_{xx})}{\partial x} + \frac{\partial(\rho uv - \tau_{xy})}{\partial y} + \frac{\partial(\rho uw - \tau_{xz})}{\partial z} = 0 \quad (4.2)$$

Y momentum:

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv - \tau_{xy})}{\partial x} + \frac{\partial(\rho v^2 + p - \tau_{yy})}{\partial y} + \frac{\partial(\rho vw - \tau_{yz})}{\partial z} = 0 \quad (4.3)$$

Z momentum:

$$\frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho uw - \tau_{xz})}{\partial x} + \frac{\partial(\rho vw - \tau_{yz})}{\partial y} + \frac{\partial(\rho w^2 + p - \tau_{zz})}{\partial z} = 0 \quad (4.4)$$

The  $\tau$  terms in these equation represent viscous stresses in a particular plane. Assuming that the fluid in question is Newtonian (i.e. stress is linearly related to rate of strain), the viscous stress terms are as follows:

$$\tau_{xx} = \frac{2}{3}\mu \left( 2\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} - \frac{\partial w}{\partial z} \right) \quad (4.5)$$

$$\tau_{xy} = \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \quad (4.6)$$

$$\tau_{xz} = \mu \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) \quad (4.7)$$

$$\tau_{yy} = \frac{2}{3}\mu \left( -\frac{\partial u}{\partial x} + 2\frac{\partial v}{\partial y} - \frac{\partial w}{\partial z} \right) \quad (4.8)$$

$$\tau_{yz} = \mu \left( \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right) \quad (4.9)$$

$$\tau_{zz} = \frac{2}{3}\mu \left( -\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} + 2\frac{\partial w}{\partial z} \right) \quad (4.10)$$

#### 4.3.4 Conservation of Energy

In accordance with the First Law of Thermodynamics, energy must also be conserved within a control volume. This is the basis for the energy equation, which describes the relationship between the time rate of change of energy in a system to the work done by forces acting on the system and the heat added to the system. Assuming that body forces (i.e. gravity) may be neglected, the conservative form of the energy equation is

$$\begin{aligned} \frac{\partial(\rho E_t)}{\partial t} + \frac{\partial}{\partial x} [(\rho E_t + p)u - u\tau_{xx} - v\tau_{xy} - w\tau_{xz} + q_x] \\ + \frac{\partial}{\partial y} [(\rho E_t + p)v - u\tau_{xy} - v\tau_{yy} - w\tau_{yz} + q_y] \\ + \frac{\partial}{\partial z} [(\rho E_t + p)w - u\tau_{xz} - v\tau_{yz} - w\tau_{zz} + q_z] = 0 \end{aligned} \quad (4.11)$$

The total energy per unit mass,  $E_t$ , is a function of both the internal energy per unit mass,  $e$ , and the kinetic energy per unit mass:

### 4.3.6 Vector Form of the Conservation Equations

When applying a finite difference method to a complex set of equations, it greatly simplifies the implementation if the equations can be rewritten in a more compact form. Following the example of Anderson, Tannehill, and Pletcher [2], the five partial differential equations shown above can be rewritten in vector form:

$$\frac{\partial \vec{U}}{\partial t} + \frac{\partial \vec{E}}{\partial x} + \frac{\partial \vec{F}}{\partial y} + \frac{\partial \vec{G}}{\partial z} = \vec{0} \quad (4.18)$$

In this equation,  $\vec{U}$  is typically referred to as the dependent variable vector:

$$\vec{U} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E_t \end{bmatrix} \quad (4.19)$$

The vectors  $\vec{E}$ ,  $\vec{F}$ , and  $\vec{G}$  represent the fluxes in the X, Y, and Z directions, respectively:

X fluxes:

$$\vec{E} = \begin{bmatrix} \rho u \\ \rho u^2 + p - \tau_{xx} \\ \rho uv - \tau_{xy} \\ \rho uw - \tau_{xz} \\ (\rho E_t + p)u - u\tau_{xx} - v\tau_{xy} - w\tau_{xz} + q_x \end{bmatrix} \quad (4.20)$$

Y fluxes:

$$\vec{F} = \begin{bmatrix} \rho v \\ \rho uv - \tau_{xy} \\ \rho v^2 + p - \tau_{yy} \\ \rho vw - \tau_{yz} \\ (\rho E_t + p)v - u\tau_{xy} - v\tau_{yy} - w\tau_{yz} + q_y \end{bmatrix} \quad (4.21)$$

Z fluxes:

$$\vec{G} = \begin{bmatrix} \rho w \\ \rho uw - \tau_{xz} \\ \rho vw - \tau_{yz} \\ \rho w^2 + p - \tau_{zz} \\ (\rho E_t + p)w - u\tau_{xz} - v\tau_{yz} - w\tau_{zz} + q_z \end{bmatrix} \quad (4.22)$$

$$E_t = \left( e + \frac{\vec{u} \cdot \vec{u}}{2} \right) \quad (4.12)$$

The heat flux vector,  $\vec{q}$ , is a function of temperature,  $T$ :

$$\vec{q} = -k \vec{\nabla} T \quad (4.13)$$

### 4.3.5 Equations of State

While the above equations describe the kinematic state of a fluid, they do not represent a closed system of equations because they do not describe the thermodynamic state of the system. To close the system requires two state equations which relate the pressure,  $p$ , and internal energy,  $e$ , to the temperature,  $T$ . For a perfect gas, these equations of state are

$$p = \rho R T \quad (4.14)$$

$$e = c_v T = \frac{RT}{\gamma - 1} \quad (4.15)$$

Here  $R$  is the gas constant, related to the specific heats  $c_p$  and  $c_v$  and their ratio  $\gamma$ :

$$\gamma = \frac{c_p}{c_v} \quad (4.16)$$

$$R = c_p - c_v = c_v (\gamma - 1) \quad (4.17)$$

### 4.3.7 Coordinate Transformation

While the grid described in Section 4.2 was expressed in terms of Cartesian coordinates, for purposes of implementation it is much simpler to treat the equations in a body-fitted curvilinear coordinate system. To allow this, the governing equations must be transformed from the Cartesian coordinates  $[x, y, z]$  of physical space into the curvilinear coordinates  $[\xi, \eta, \zeta]$  of computational space. Derivatives of an arbitrary function  $f$  in Cartesian coordinates can be transformed in curvilinear coordinates by a simple application of the “Chain Rule” of differential calculus:

$$\frac{\partial f}{\partial x} = \frac{\partial f}{\partial \xi} \frac{\partial \xi}{\partial x} + \frac{\partial f}{\partial \eta} \frac{\partial \eta}{\partial x} + \frac{\partial f}{\partial \zeta} \frac{\partial \zeta}{\partial x} \quad (4.23)$$

$$\frac{\partial f}{\partial y} = \frac{\partial f}{\partial \xi} \frac{\partial \xi}{\partial y} + \frac{\partial f}{\partial \eta} \frac{\partial \eta}{\partial y} + \frac{\partial f}{\partial \zeta} \frac{\partial \zeta}{\partial y} \quad (4.24)$$

$$\frac{\partial f}{\partial z} = \frac{\partial f}{\partial \xi} \frac{\partial \xi}{\partial z} + \frac{\partial f}{\partial \eta} \frac{\partial \eta}{\partial z} + \frac{\partial f}{\partial \zeta} \frac{\partial \zeta}{\partial z} \quad (4.25)$$

Using this approach, the vector form of the Navier-Stokes equations shown in Equation 4.18 can be transformed into curvilinear coordinates as shown in Equation 4.26. Here and in the rest of this section, a subscript indicates a partial derivative with respect to the subscripted coordinate.

$$\vec{U}_t + \vec{E}_\xi \xi_x + \vec{E}_\eta \eta_x + \vec{E}_\zeta \zeta_x + \vec{F}_\xi \xi_y + \vec{F}_\eta \eta_y + \vec{F}_\zeta \zeta_y + \vec{G}_\xi \xi_z + \vec{G}_\eta \eta_z + \vec{G}_\zeta \zeta_z = \vec{0} \quad (4.26)$$

The solution of Equation 4.26 requires that the nine metric terms  $\xi_x, \xi_y, \xi_z, \eta_x, \eta_y, \eta_z, \zeta_x, \zeta_y,$  and  $\zeta_z$  be defined at every point on the grid. These metrics can be defined in terms of the Jacobian of the transformation:

$$J = \frac{\partial(\xi, \eta, \zeta)}{\partial(x, y, z)} = \begin{vmatrix} \xi_x & \xi_y & \xi_z \\ \eta_x & \eta_y & \eta_z \\ \zeta_x & \zeta_y & \zeta_z \end{vmatrix} \quad (4.27)$$

$$J = \frac{1}{J^{-1}} = \frac{1}{\frac{\partial(x, y, z)}{\partial(\xi, \eta, \zeta)}} = \frac{1}{\begin{vmatrix} x_\xi & x_\eta & x_\zeta \\ y_\xi & y_\eta & y_\zeta \\ z_\xi & z_\eta & z_\zeta \end{vmatrix}} \quad (4.28)$$

$$\begin{aligned} \xi_x &= J(y_\eta z_\zeta - y_\zeta z_\eta) \\ \xi_y &= J(x_\zeta z_\eta - x_\eta z_\zeta) \\ \xi_z &= J(x_\eta y_\zeta - x_\zeta y_\eta) \\ \eta_x &= J(y_\zeta z_\xi - y_\xi z_\zeta) \\ \eta_y &= J(x_\xi z_\zeta - x_\zeta z_\xi) \\ \eta_z &= J(x_\zeta y_\xi - x_\xi y_\zeta) \\ \zeta_x &= J(y_\xi z_\eta - y_\eta z_\xi) \\ \zeta_y &= J(x_\eta z_\xi - x_\xi z_\eta) \\ \zeta_z &= J(x_\xi y_\eta - x_\eta y_\xi) \end{aligned} \quad (4.29)$$

While this transformation works for the majority of the grid, it fails at the center line. The reason for the failure of the transformation at the center line is that the transformation requires a one-to-one mapping of points between physical and computational spaces; this does not hold at the center line, where 38 points in computational space map to a single point in physical space. As a result, the transformation at the center line must be treated as a special case; the  $x_\eta$ ,  $y_\eta$ , and  $z_\eta$  metrics on the center line are forced to be equal to their respective values at the first point out radially

from the center line at the same axial and azimuthal location. The center line also requires special treatment in boundary conditions, as described in Section 4.5.1.

There are several ways in which the mapping difficulty at the center line could be handled other than that described above, although none of them were investigated for this study. Most of these methods involve removing the center line as a boundary in some fashion. For instance, the grid could be treated in such a way that opposing half-planes (such as the  $K=1$  and  $K=19$  planes) were treated as the same coordinate plane. Alternatively, the domain could be regridded using a rectangular arrangement rather than the cylindrical arrangement used in this study. The latter of these is attractive because it would do away with the center line entirely, but it would also make direct comparisons between the numerical results of this study and those of Khavaran, Kresja, and Kim [11, 12] more difficult.

## 4.4 Numerical Method

The numerical method used in this study to approximate the partial differential equations shown in Equation 4.18 is the MacCormack explicit scheme [2]. The MacCormack scheme is a variation of the Lax-Wendroff family of two step schemes. It is second order accurate in both time and space. The MacCormack scheme has proven itself to be quite robust over a wide range of problems; in particular, Scott [22, 23, 25], Rozmajzl [19], and al Qadi [1] have used this method successfully to simulate unsteady jet flows in both two dimensional rectangular and axisymmetric jets. The MacCormack scheme is also well known for providing good resolution of discontinuities such as shock waves.

The MacCormack method is a two step predictor/corrector scheme. In the predictor step, an intermediate solution  $\vec{U}^{n+1}$  is computed by

$$\begin{aligned}\vec{U}_{i,j,k}^{n+1} = & \vec{U}_{i,j,k}^n - \frac{\Delta t}{\Delta x} (\vec{E}_{i+1,j,k}^n - \vec{E}_{i,j,k}^n) \\ & - \frac{\Delta t}{\Delta y} (\vec{F}_{i,j+1,k}^n - \vec{F}_{i,j,k}^n) - \frac{\Delta t}{\Delta z} (\vec{G}_{i,j,k+1}^n - \vec{G}_{i,j,k}^n)\end{aligned}\quad (4.30)$$

In the corrector step, Lax averaging is used on the previous time step's solution  $\vec{U}^n$  and the intermediate solution  $\vec{U}^{n+1}$  in the temporal difference used to compute the new solution  $\vec{U}^{n+1}$ , as shown in Equation 4.31. This effectively results in the averaging of two first order accurate solutions, one involving only forward differences and the other involving only backward differences.

$$\begin{aligned}\vec{U}_{i,j,k}^{n+1} = & \frac{1}{2} \left[ \vec{U}_{i,j,k}^n + \vec{U}_{i,j,k}^{n+1} - \frac{\Delta t}{\Delta x} (\vec{E}_{i,j,k}^{n+1} - \vec{E}_{i-1,j,k}^{n+1}) \right. \\ & \left. - \frac{\Delta t}{\Delta y} (\vec{F}_{i,j,k}^{n+1} - \vec{F}_{i,j-1,k}^{n+1}) - \frac{\Delta t}{\Delta z} (\vec{G}_{i,j,k}^{n+1} - \vec{G}_{i,j,k-1}^{n+1}) \right]\end{aligned}\quad (4.31)$$

The direction of differencing used to compute derivative terms such as pressure gradient, viscous stresses, and heat flux in the flux vectors  $\vec{E}$ ,  $\vec{F}$  and  $\vec{G}$  is important in retaining the second order accuracy of the scheme. Differences used to compute these derivatives of the primitive variables must be in the opposite direction of the differencing of the flux vector if they are in the same coordinate. This is done to maintain second order accuracy in space [2]. For instance, if the  $x$  differencing of  $\vec{E}$  is in the forward direction, then any  $x$  derivatives used in composing  $\vec{E}$  must be computed using backward differences. Cross derivatives, such as  $y$  and  $z$  derivatives used in composing  $\vec{E}$ , are computed using central differences.

The predictor and corrector equations shown in Equations 4.30 and 4.31 assume that the  $i$ ,  $j$ , and  $k$  indices are aligned with the Cartesian axes. However, such is not the case when working with the grid described in Section 4.2. In order to account for this, the coordinate transformation described in Section 4.3.7 must be applied to Equations 4.30 and 4.31. The transformed versions of Equations 4.30 and 4.31 are

Predictor:

$$\begin{aligned}
\bar{U}_{i,j,k}^{n+1} = & \bar{U}_{i,j,k}^n - \frac{\Delta t}{\Delta \xi} (\bar{E}_{i,j,k}^n - \bar{E}_{i-1,j,k}^n) (\xi_x)_{i,j,k} - \frac{\Delta t}{\Delta \eta} (\bar{E}_{i,j,k}^n - \bar{E}_{i,j-1,k}^n) (\eta_x)_{i,j,k} \\
& - \frac{\Delta t}{\Delta \zeta} (\bar{E}_{i,j,k}^n - \bar{E}_{i,j,k-1}^n) (\zeta_x)_{i,j,k} - \frac{\Delta t}{\Delta \xi} (\bar{F}_{i,j,k}^n - \bar{F}_{i-1,j,k}^n) (\xi_y)_{i,j,k} \\
& - \frac{\Delta t}{\Delta \eta} (\bar{F}_{i,j,k}^n - \bar{F}_{i,j-1,k}^n) (\eta_y)_{i,j,k} - \frac{\Delta t}{\Delta \zeta} (\bar{F}_{i,j,k}^n - \bar{f}_{i,j,k-1}^n) (\zeta_y)_{i,j,k} \\
& - \frac{\Delta t}{\Delta \xi} (\bar{G}_{i,j,k}^n - \bar{G}_{i-1,j,k}^n) (\xi_z)_{i,j,k} - \frac{\Delta t}{\Delta \eta} (\bar{G}_{i,j,k}^n - \bar{G}_{i,j-1,k}^n) (\zeta_z)_{i,j,k} \\
& - \frac{\Delta t}{\Delta \zeta} (\bar{G}_{i,j,k}^n - \bar{G}_{i,j,k-1}^n) (\zeta_z)_{i,j,k}
\end{aligned} \tag{4.32}$$

Corrector:

$$\begin{aligned}
\bar{U}_{i,j,k}^{n+1} = & \frac{1}{2} \left[ \bar{U}_{i,j,k}^n + \bar{U}_{i,j,k}^{n+1} \right. \\
& - \frac{\Delta t}{\Delta \xi} (\bar{E}_{i+1,j,k}^{n+1} - \bar{E}_{i,j,k}^{n+1}) (\xi_x)_{i,j,k} - \frac{\Delta t}{\Delta \eta} (\bar{E}_{i,j+1,k}^{n+1} - \bar{E}_{i,j,k}^{n+1}) (\eta_x)_{i,j,k} \\
& - \frac{\Delta t}{\Delta \zeta} (\bar{E}_{i,j,k+1}^{n+1} - \bar{E}_{i,j,k}^{n+1}) (\zeta_x)_{i,j,k} - \frac{\Delta t}{\Delta \xi} (\bar{F}_{i+1,j,k}^{n+1} - \bar{F}_{i,j,k}^{n+1}) (\xi_y)_{i,j,k} \\
& - \frac{\Delta t}{\Delta \eta} (\bar{F}_{i,j+1,k}^{n+1} - \bar{F}_{i,j,k}^{n+1}) (\eta_y)_{i,j,k} - \frac{\Delta t}{\Delta \zeta} (\bar{F}_{i,j,k+1}^{n+1} - \bar{f}_{i,j,k}^{n+1}) (\zeta_y)_{i,j,k} \\
& - \frac{\Delta t}{\Delta \xi} (\bar{G}_{i+1,j,k}^{n+1} - \bar{G}_{i,j,k}^{n+1}) (\xi_z)_{i,j,k} - \frac{\Delta t}{\Delta \eta} (\bar{G}_{i,j+1,k}^{n+1} - \bar{G}_{i,j,k}^{n+1}) (\zeta_z)_{i,j,k} \\
& \left. - \frac{\Delta t}{\Delta \zeta} (\bar{G}_{i,j,k+1}^{n+1} - \bar{G}_{i,j,k}^{n+1}) (\zeta_z)_{i,j,k} \right]
\end{aligned} \tag{4.33}$$

#### 4.4.1 Stability and Accuracy

A numerical scheme is said to be stable if errors in the solution do not grow as the solution is advanced in time. For most numerical schemes, this results in a relation

between the mesh size and the maximum allowable time step size. The relationship can be found directly using von Neumann stability analysis for linear equations, but not for highly nonlinear equations such as the Navier-Stokes equations [24]. However, for the MacCormack scheme as applied to the Navier-Stokes equations, the following empirical formulation [2] may be used to compute a global time step:

$$\Delta t \leq \frac{\sigma (\Delta t)_{CFL}}{1 + \frac{2}{Re_{\Delta}}} \quad (4.34)$$

In Equation 4.34, the term  $\sigma$  is a safety factor, typically on the order of 0.9 or less. The time step  $(\Delta t)_{CFL}$  is that defined by the inviscid CFL condition, namely

$$(\Delta t)_{CFL} \leq \left( \frac{|u|}{\Delta x} + \frac{|v|}{\Delta y} + \frac{|w|}{\Delta z} + a \sqrt{\frac{1}{(\Delta x)^2} + \frac{1}{(\Delta y)^2} + \frac{1}{(\Delta z)^2}} \right)^{-1} \quad (4.35)$$

The cell Reynolds number  $Re_{\Delta}$  is the minimum of the cell Reynolds numbers, given by

$$Re_{\Delta} = \min \left( \frac{\rho |u| \Delta x}{\mu}, \frac{\rho |v| \Delta y}{\mu}, \frac{\rho |w| \Delta z}{\mu} \right) \quad (4.36)$$

The local speed of sound  $a$  is computed using the usual isentropic relation [10]:

$$a = \sqrt{\gamma \frac{p}{\rho}} = \sqrt{\gamma RT} \quad (4.37)$$

For every time step, a local time step is computed at every grid point using Equation 4.34, and the global minimum value of  $\Delta t$  is used to advance the solution in time.

The accuracy of a numerical scheme is generally expressed in terms of the order of the leading terms in the truncation error. The MacCormack explicit scheme used

in this study is said to be second order accurate in both time and space. This means that the leading terms in the truncation error include factors of  $(\Delta t)^2$ ,  $(\Delta x)^2$ ,  $(\Delta y)^2$ , or  $(\Delta z)^2$ . Each term in the truncation error represents either an error in either the phase of the numerical solution (a dispersion error) or the magnitude of the numerical solution (a dissipation error). Dispersion error terms contain odd derivatives of one or more of the dependent variables, while dissipation error terms contain even derivatives of one or more dependent variables. The leading term in the truncation error of the MacCormack explicit scheme contain odd spatial derivatives, and thus it is said to be dispersive. More details on the stability and accuracy of the MacCormack explicit scheme are given by Scott [24] and Rozmajzl [19].

#### 4.4.2 Damping

One of the most noticeable effects of the dispersion described above is the tendency of the numerical solution to oscillate or “ring” near discontinuities or strong gradients, such as those associated with shock waves. In an attempt to minimize this effect, an artificial viscosity scheme using fourth order pressure damping is added to the numerical solution method. This damping scheme introduces additional dissipation in areas of high pressure gradients, such as shock waves. The form of this pressure damping scheme is given by

$$\bar{D}_x = \epsilon_e (\Delta x)^4 \frac{\partial}{\partial x} \left[ \frac{|u| + a}{4p} \left| \frac{\partial^2 p}{\partial x^2} \right| \frac{\partial \vec{U}}{\partial x} \right] \quad (4.38)$$

$$\bar{D}_y = \epsilon_e (\Delta y)^4 \frac{\partial}{\partial y} \left[ \frac{|v| + a}{4p} \left| \frac{\partial^2 p}{\partial y^2} \right| \frac{\partial \vec{U}}{\partial y} \right] \quad (4.39)$$

$$\vec{D}_z = \epsilon_e (\Delta z)^4 \frac{\partial}{\partial z} \left[ \frac{|w| + a}{4p} \left| \frac{\partial^2 p}{\partial z^2} \right| \frac{\partial \vec{U}}{\partial z} \right] \quad (4.40)$$

The damping vectors  $\vec{D}_x$ ,  $\vec{D}_y$ , and  $\vec{D}_z$  are added to the dependent variable vector  $\vec{U}$  during each time step.

## 4.5 Boundary Conditions

One of the most difficult issues facing the numerical simulation of jet noise, and indeed computational fluid dynamics in general, is the formulation and implementation of physically meaningful boundary conditions for unsteady flow. The way in which boundary conditions are set has a profound impact on the behavior of the flow in the interior of the domain. The boundary conditions must be formulated to accurately model the physics of the flow at each of the boundaries of the computational domain, so that the solution on the interior of the domain accurately reflects the physics of the flow as a whole.

### 4.5.1 Center Line Boundary Conditions

The boundary conditions imposed at the center line ( $J=1$ ) are at once the simplest and the most problematic in the computation. In a typical axisymmetric calculation, the center line condition is formulated such that no flux of mass, momentum, or energy is allowed across the center line, resulting in

$$\left. \frac{\partial \vec{U}}{\partial r} \right|_{r=0} = \vec{0} \quad (4.41)$$

In most cases, this condition is implemented using a simple first-order accurate forward difference, so that  $\vec{U}_{i,1}$  is set equal to  $\vec{U}_{i,2}$  for all values of  $i$  (assuming that  $i$

is the index in the axial direction and  $j$  is the index in the radial direction). However, in a three dimensional calculation, the assumption that fluxes are not allowed to cross the centerline (and thus Equation 4.41) does not necessarily hold. In this study, there is an additional complication in that there are a total of 38 points in computational space which map to a single point in physical space along the center line, as discussed in Section 4.3.7. To allow fluxes to cross the center line as well as to address this complication in the grid mapping, the center line boundary condition assigns to the centerline the average of the values of  $\vec{U}$  at all the surrounding points at the same axial distance:

$$\vec{U}_{i,1,k} = \frac{1}{36} \sum_{n=2}^{37} \vec{U}_{i,2,n}, \forall i \in [2, 140], k \in [2 : 37] \quad (4.42)$$

However, upon running the simulation with this boundary condition implemented as shown above, it was found that this boundary condition caused some undesirable effects near the center line. The first and most bothersome of these was the introduction of a strong azimuthal swirl near the center line. Another undesirable effect of this condition is the tendency of pressure disturbances to be reflected strongly from the center line, resulting in oscillations in contours of constant pressure and Mach number near the center line. To counteract these effects, a second order accurate forward difference was substituted in one of the later simulation runs for the first order difference used above. This led to the implementation of the center line boundary condition expressed as

$$\vec{U}_{i,1,k} = \frac{1}{36} \sum_{n=2}^{37} \frac{4\vec{U}_{i,2,n} - \vec{U}_{i,3,n}}{3}, \forall i \in [2, 140], k \in [2 : 37] \quad (4.43)$$

The second order accurate form of this boundary condition caused less reflection than the first order form, although it did not suppress the reflections entirely. As a result, formulation and implementation of a non-reflective center line boundary condition remains an area of ongoing research.

#### 4.5.2 Far Field Boundary Conditions

For this study, it was assumed that the computational domain was enclosed by a large cylindrical wall at its outer radial extent ( $J=61$ ). As such, this far field boundary was treated as a viscous wall, which requires no slip, a fixed wall temperature, and a zero normal pressure gradient. These mathematical conditions are:

$$\begin{aligned}\vec{u}|_{r=r_{max}} &= \vec{0} \\ T|_{r=r_{max}} &= T_{w,2} \\ \frac{\partial p}{\partial n}|_{r=r_{max}} &= 0\end{aligned}\tag{4.44}$$

These conditions are applied to the dependent variable vector  $\vec{U}$  in the following fashion. To satisfy the no slip condition, all three components of momentum are set to zero. Density is computed by calculating the pressure at one point off the boundary in the radial direction, then using that value to solve Equation 4.45 using the fixed wall temperature  $T_{w,2}$ . This equation is a combination of Equations 4.14 and 4.15.

$$\rho = \frac{p}{(\gamma - 1)c_v T}\tag{4.45}$$

Using the density obtained from Equation 4.45, the energy term may be computed using Equation 4.12. This leads to the implementation of these boundary conditions as

$$\vec{U}_{i,61,k} = \begin{bmatrix} \frac{(p)_{i,60,k}}{(\gamma-1)c_v T_{w,2}} \\ 0 \\ 0 \\ 0 \\ (\rho)_{i,61,k} c_v T_{w,2} \end{bmatrix} \quad \forall i \in [2 : 140], k \in [2 : 37] \quad (4.46)$$

### 4.5.3 Inflow Boundary Conditions

The inflow boundary ( $I=1$ ) may be divided into three segments. The first of these segments ( $I=1, J=[1:24]$ ) represents the area through which flow enters the nozzle, while the second ( $I=1, J=[27:61]$ ) represents the area through which the freestream flow enters the computational domain. Both of these regions are areas of subsonic inflow. The third segment of the inflow boundary ( $I=1, J=[25:26]$ ) represents the intersection of the inflow boundary with the nozzle wall and is treated in the nozzle wall boundary conditions discussed in Section 4.5.5.

Before discussing the details of the inflow boundary conditions imposed on the first two segments of the inflow boundary described above, a brief diversion into characteristic analysis is required. In a seminal paper on the subject of time-dependent boundary conditions for hyperbolic systems of partial differential equations, Thompson [29] showed that the Navier-Stokes equations may be rewritten in terms of characteristic wave quantities. These characteristic quantities are

$$\begin{aligned} \mathcal{L}_1 &= \lambda_1 \left( \frac{\partial p}{\partial x} - \rho a \frac{\partial u}{\partial x} \right) \\ \mathcal{L}_2 &= \lambda_2 \left( a^2 \frac{\partial \rho}{\partial x} - \frac{\partial p}{\partial x} \right) \\ \mathcal{L}_3 &= \lambda_3 \frac{\partial v}{\partial x} \\ \mathcal{L}_4 &= \lambda_4 \frac{\partial w}{\partial x} \end{aligned} \quad (4.47)$$

$$\mathcal{L}_5 = \lambda_5 \left( \frac{\partial p}{\partial x} + \rho a \frac{\partial u}{\partial x} \right)$$

Here, the coefficients  $\lambda_i$  are speeds at which the waves represented by  $\mathcal{L}_i$  travel in the X direction. The quantities  $\lambda_1$  and  $\lambda_5$  are the sound speeds for the left-running and right-running acoustic waves respectively. Similarly,  $\lambda_3$  and  $\lambda_4$  are the speeds at which vorticity components in the Z and Y directions respectively are advected in the X direction. Finally,  $\lambda_2$  is the speed at which entropy is advected in the X direction. Thus, there are a total of five waves to be considered: two acoustic waves, two vorticity waves, and an entropy wave. The direction of these waves at a boundary is of particular interest in the formulation and implementation of time-dependent boundary conditions, as Thompson showed that the number of waves traveling out of a domain at a boundary is equal to the number of boundary conditions which must be specified on that boundary.

In the characteristic analysis described above, Thompson showed that at a subsonic inflow boundary four of the five characteristic waves present (the right-running acoustic wave, the entropy wave, and two vorticity waves) travel into the domain, while the fifth (the left-running acoustic wave) travels out of the domain. Thus, four quantities must be specified at the inflow boundary while a fifth must be extrapolated. The quantity selected for extrapolation is the axial velocity component  $u$ , while the fixed quantities are the stagnation temperature  $T^0$ , the stagnation pressure  $p^0$ , and the velocity components  $v$  and  $w$ .

In the first segment of the inflow boundary, that lying within the interior of the nozzle, the conditions imposed upon the flow are

$$\begin{aligned}
\frac{\partial u}{\partial x} &= 0 \\
v &= 0 \\
w &= 0 \\
p^0 &= p_1^0 \\
T^0 &= T_1^0
\end{aligned} \tag{4.48}$$

A radial Mach number distribution is also assumed to account for the presence of an initial boundary layer profile at the inlet of the nozzle; this is used primarily to compute the static temperature  $T_j$  and static pressure  $p_j$  from the stagnation temperature  $T_1^0$  and stagnation pressure  $p_1^0$  using the usual isentropic relations:

$$T_j = \frac{T_1^0}{1 + \frac{\gamma-1}{2} M_j^2} \tag{4.49}$$

$$p_j = \frac{p_1^0}{1 + \frac{\gamma-1}{2} M_j^2}^{\frac{\gamma}{\gamma-1}} \tag{4.50}$$

Using the static temperature and pressure, a density may be computed using Equation 4.45. The dependent variable vector  $\vec{U}$  may then be calculated by

$$\vec{U}_{1,j,k} = \begin{bmatrix} \frac{p_j}{(\gamma-1)c_v T_j} \\ (\rho)_{1,j,k} |u|_{2,j,k} \\ 0 \\ 0 \\ (\rho)_{1,j,k} \left( c_v T_j + \frac{u_{2,j,k}^2}{2} \right) \end{bmatrix} \forall j \in [1 : 24], k \in [2 : 37] \tag{4.51}$$

The absolute value of  $u_{2,j,k}$  is used in the  $x$  momentum term of  $\vec{U}_{1,j,k}$  so that the flow is constrained to be in the positive  $x$  (i.e. streamwise) direction at the inflow plane.

In the second portion of the inflow boundary, that lying outside the nozzle, the conditions are imposed are those shown in Equation 4.53.

$$\begin{aligned}
\frac{\partial u}{\partial x} &= 0 \\
v &= 0 \\
w &= 0 \\
p^0 &= p_2^0 \\
T^0 &= T_2^0
\end{aligned} \tag{4.52}$$

These are essentially identical to the conditions found within the nozzle, except that different values for the stagnation pressure  $p^0$  and stagnation temperature  $T^0$  are used. Also, a constant Mach number  $M_{\infty,2}$  is assumed rather than a varying distribution. The implementation form of the boundary conditions used outside the nozzle is also very similar to that used within the nozzle; it is given by Equation 4.53.

$$\vec{U}_{1,j,k} = \begin{bmatrix} \frac{p_j}{(\gamma-1)c_v T_j} \\ (\rho)_{1,j,k} (u)_{2,j,k} \\ 0 \\ 0 \\ (\rho)_{1,j,k} \left( c_v T_j + \frac{u_{2,j,k}^2}{2} \right) \end{bmatrix} \forall j \in [27 : 61], k \in [2 : 37] \tag{4.53}$$

#### 4.5.4 Outflow Boundary Conditions

The boundary conditions imposed on the outflow boundary (I=141) are quite different from those at the the inflow. The flow at this boundary is nominally subsonic, like that at the inflow boundary. Thompson's characteristic analysis [29] showed that in the case of subsonic outflow, four waves (the right-running acoustic wave and the

entropy wave, and two vorticity waves) travel out of the domain from the boundary while the fifth (the left-running acoustic wave) travels into the domain through the boundary. Thus, the outflow condition must fix one of the primitive variables and extrapolate the other four from the interior. The primitive variable selected to be fixed on this boundary is the static pressure  $p$ , while the variables selected for extrapolation are the density  $\rho$  and the velocity components  $u$ ,  $v$ , and  $w$ . These conditions are

$$\begin{aligned}
\frac{\partial \rho}{\partial x} &= 0 \\
\frac{\partial u}{\partial x} &= 0 \\
\frac{\partial v}{\partial x} &= 0 \\
\frac{\partial w}{\partial x} &= 0 \\
p &= 0.98p_{\infty,2}
\end{aligned} \tag{4.54}$$

The condition on the static pressure imposes a small negative pressure gradient over the length of the computational domain, so that the flow will tend to be primarily in the positive X direction.

However, imposing a fixed pressure across the entire outflow boundary creates a problem with the physics of the flow. A fixed pressure over the entire boundary would cause any pressure disturbances propagating downstream to be reflected back into the computational domain rather than pass out of the domain. However, the opposite extreme of extrapolating the static pressure over the entire boundary will tend to accelerate the flow to supersonic speeds. One solution used in this study for this problem is to fix the pressure on a small portion of the boundary where pressure disturbances are not expected to propagate downstream and extrapolate the pressure over the rest of the boundary:

$$p_{141,j,k} = \begin{cases} 0.98p_{\infty,2} & j \in [1 : 15] \\ p_{140,j,k} & j > 15 \end{cases}, k \in [2 : 37] \quad (4.55)$$

The selection of this particular  $j$  range for the fixed pressure condition is strictly arbitrary, based upon experience and the behavior of the solver code. Fixing the pressure near the center line, where the flow speed is expected to be highest, encourages the flow to decelerate to subsonic speeds as it approaches the outflow boundary.

Using the relation for static pressure shown in Equation 4.55, the outflow boundary condition may be expressed as

$$\vec{U}_{141,j,k} = \begin{bmatrix} (\rho)_{140,j,k} \\ (\rho)_{140,j,k} (u)_{140,j,k} \\ (\rho)_{140,j,k} (v)_{140,j,k} \\ (\rho)_{140,j,k} (w)_{140,j,k} \\ \frac{p_{141,j,k}}{\gamma-1} + (\rho)_{140,j,k} \frac{(u)_{140,j,k}^2 + (v)_{140,j,k}^2 + (w)_{140,j,k}^2}{2} \end{bmatrix} \quad \forall j \in [1 : 61], k \in [2 : 37] \quad (4.56)$$

Another solution to the problem of non-physical reflections from a fixed pressure boundary is to extrapolate the pressure from the interior of the flow over the entire area of the boundary by assuming that the axial pressure gradient  $\frac{\partial p}{\partial x}$  is equal to zero. This assumes that pressure disturbances are small by the time they reach the outflow boundary, but it guarantees that disturbances are allowed to propagate out of the computational domain. Unfortunately, this also is mathematically identical to a supersonic outflow condition, and it has the potential to drive a flow that would be subsonic at the outflow boundary in an experiment to be supersonic in the computation. Thus, this approach requires substantial grid stretching near the outflow boundary. This boundary condition is

$$\vec{U}_{141,j,k} = \begin{bmatrix} (\rho)_{140,j,k} \\ (\rho)_{140,j,k} (u)_{140,j,k} \\ (\rho)_{140,j,k} (v)_{140,j,k} \\ (\rho)_{140,j,k} (w)_{140,j,k} \\ \frac{p_{140,j,k}}{\gamma-1} + (\rho)_{140,j,k} \frac{(u)_{140,j,k}^2 + (v)_{140,j,k}^2 + (w)_{140,j,k}^2}{2} \end{bmatrix} \quad \forall j \in [1 : 61], k \in [2 : 37] \quad (4.57)$$

#### 4.5.5 Nozzle Wall Boundary Conditions

The boundary conditions on the nozzle walls ( $I=[1:50]$ ,  $J=[25:26]$ ) are the same viscous wall conditions described in Section 4.5.2 for the far field boundary and quite similar to those summarized in Equation 4.44. The only differences between the implementation of the viscous wall conditions on the nozzle wall and those on the far field boundary involve the direction in which the pressure gradient is computed (for the outer surface only) and the treatment of the surface temperature. The inner surface of the nozzle is assumed to be at  $T_{w,1}$ , while the outer surface is assumed to be at  $T_{w,2}$ . This assumption of constant wall temperature is based upon observations made in experimental tests. The boundary conditions for the inner and outer nozzle surfaces are

Inner nozzle surface:

$$\vec{U}_{i,25,k} = \begin{bmatrix} \frac{(p)_{i,24,k}}{(\gamma-1)c_v T_{w,1}} \\ 0 \\ 0 \\ 0 \\ (\rho)_{i,25,k} c_v T_{w,1} \end{bmatrix} \quad \forall i \in [1 : 50], k \in [2 : 37] \quad (4.58)$$

Outer nozzle surface:

$$\vec{U}_{i,26,k} = \begin{bmatrix} \frac{(p)_{i,27,k}}{(\gamma-1)c_v T_{w,2}} \\ 0 \\ 0 \\ 0 \\ (\rho)_{i,26,k} c_v T_{w,2} \end{bmatrix} \quad \forall i \in [1 : 50], k \in [2 : 37] \quad (4.59)$$

### 4.5.6 Periodic Boundary Conditions

As mentioned in Section 4.2, there are two overlapping grid planes in the computational domain which are used for the enforcement of a periodic boundary condition. This allows for the propagation of disturbances in a helical mode, as described in Section 2.1. To implement this condition for the K=1 plane, the values of  $\vec{U}$  computed in the K=37 plane are simply copied directly into the corresponding nodes in the K=1 plane:

$$\vec{U}_{i,j,1} = \vec{U}_{i,j,37}, \forall i \in [1 : 141], j \in [1 : 61] \quad (4.60)$$

Similarly, the values of  $\vec{U}$  in the K=2 plane are copied into the corresponding nodes in the K=38 plane:

$$\vec{U}_{i,j,38} = \vec{U}_{i,j,2}, \forall i \in [1 : 141], j \in [1 : 61] \quad (4.61)$$

## 4.6 Visualization Techniques and Tools

An important tool in the analysis of the vast amounts of results produced by numerical simulations is the visualization of numerical results using 3D computer graphics [8]. Visualization gives the analyst a graphical representation of the physical behavior of the flow represented by a numerical simulation, leading to a better understanding of the underlying physics than would be possible otherwise. In this way, visualization techniques are the computational analogues of techniques used in experimental settings such as tufts, smoke trails, and schlieren and shadowgraph photographs.

The most commonly used visualization technique in this study is contour generation. This is a technique originally developed for two dimensional simulations, but it is also quite effective for planar slices through a 3D simulation. Contour generation involves the computation of several isovalue curves in a selected scalar field, such as pressure or Mach number. Graphically, these curves can either be displayed directly or used to generate a continuous color image. Contour plots are most often used to identify and locate physical phenomena such as shock waves in a flow, and a creative choice of the scalar field plotted can lead to considerable amounts of physical insight. For instance, the Mach number and entropy fields are useful in analyzing the behavior of critical features in the shear layer, such as vortex shedding and interaction as well as their subsequent breakdown and vortex-shock interaction. Further, the pressure and density fields may be used to identify shock waves and Prandtl-Meyer expansions. One other scalar field used in the analyses of this study for the identification of shocks and expansion fans is the so-called “shock function,” shown in Equation 4.62. This function tends to be larger than unity in regions where there are both a supersonic velocity and a strong streamwise pressure gradient, such as shock waves. Additionally, the shock function will tend to be negative in regions where the flow is accelerating due to a favorable pressure gradient.

$$f_{shock} = \frac{\vec{u} \cdot \vec{\nabla} p}{a \|\vec{\nabla} p\|} \quad (4.62)$$

A second visualization technique sparingly used in this study is the technique of oriented glyphs, also known the “hedgehog” technique [21]. This involves the display of arrows to indicate the direction of a selected vector field; the nickname comes from the spiny appearance of the resulting plot. Additional information can be encoded in

a hedgehog plot by varying the length of the arrows based on the magnitude of the vector field or by varying the coloration of the vectors based on a scalar field. The hedgehog technique is much like the tufts used in some wind tunnels and on some experimental aircraft; however, unlike tufts in experimental settings, hedgehog plots do not have to be used to visualize the velocity field and may be used to examine any conceivable vector field, including the pressure gradient, vorticity, and principal stress vector fields.

The software used for the vast majority of visualizations in this study is the FAST package from NASA Ames' Numerical Aerodynamics Simulation Facility [31]. FAST is a "turnkey" visualization system designed specifically for computational fluid dynamics applications; it builds on NASA Ames' earlier PLOT3D visualization system [30] and presents the user with a fully graphical environment for the analysis of CFD results. However, FAST is tied to the specific hardware capabilities of Silicon Graphics' workstations.

Another software package used for some visualization purposes, particularly in the early stages of this study, is the Visualization Toolkit (VTK) from General Electric Medical Imaging Systems [21]. Unlike FAST, VTK is not a turnkey visualization system; rather, it is a general purpose visualization library with applications to a wide range of visualization problems. While VTK does not present the user with a graphical interface like FAST, it allows for easy scripting of repetitive tasks for the generation of animations. Furthermore, it is portable to a much wider ranges of computers than FAST.

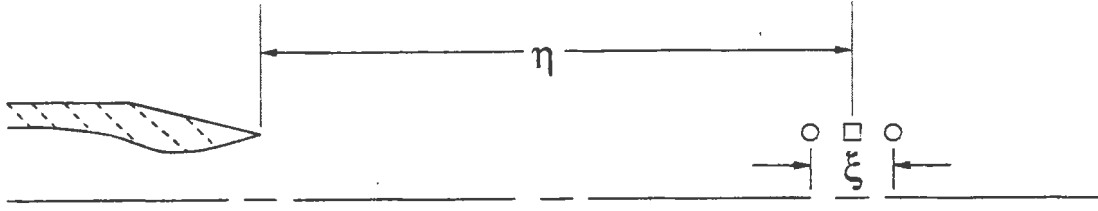


Figure 4.4: Notation for Turbulence and Cross Correlation Calculations

## 4.7 Unsteady Analysis

During each simulation run, predictions computed at many points within the flow field. These predictions may analyzed in much the same way as one would examine experimental data. The computation and analysis of unsteady quantities from the numerical simulations must done so that direct comparisons between the simulation results and experimental data may be made as simply as possible.

### 4.7.1 Computation of Unsteady Quantities

The location and types of unsteady quantities computed from the numerical simulation were chosen with great care to compare with results from either Yamamoto's experimental work [32] or Khavaran, Kresja, and Kim's numerical predictions [11, 12]. The most direct comparison between the results of this study and those cited above can be made with profiles of axial velocity  $u$  and turbulence intensity in either the radial or axial directions. For these comparisons to be made, the axial velocity component  $u$  must recorded at regular intervals in time within the numerical simulation at several points within the computational domain and stored for later analysis. The analysis techniques used for these comparisons is described in Section 4.7.2.

| $\eta/D$ | I Index |
|----------|---------|
| 0.20     | 53      |
| 0.96     | 64      |
| 2.24     | 77      |
| 4.22     | 91      |
| 8.86     | 110     |

Table 4.1: Axial Locations of Radial Velocity Profile Sampling

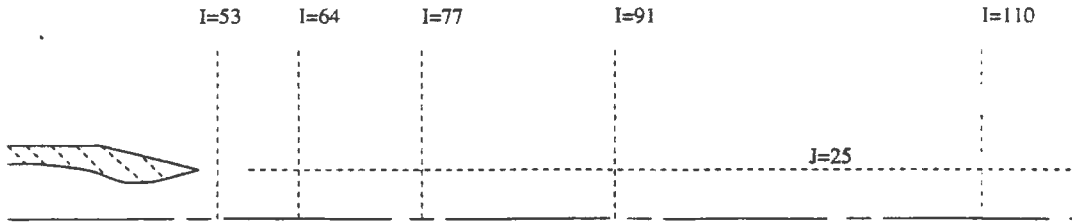


Figure 4.5: Locations of Velocity Sampling

In the case of radial profiles of velocity and turbulence intensity, both Yamamoto and Khavaran, Kresja, and Kim show these quantities at several locations. Each of these axial locations has an associated distance  $\eta$  from the exit plane of the nozzle, as shown in Figure 4.4. This distance is typically normalized by the nozzle exit diameter  $D$ . For this study, a total of five axial locations were selected; the locations in terms of  $\eta/D$  as well as the I index of the grid are shown in Table 4.1. For each of these locations,  $u$  values are recorded in the  $K=1$  plane for  $J$  values of 1 through 45 at regular intervals of fifteen time steps. The positions of these locations relative to the nozzle lip are shown with the five vertical dotted lines in Figure 4.5.

For axial turbulence intensity profiles, two lines were selected along which to sample values of  $u$ . The first of these lay along the center line, at  $J=1$ , while the second lay on the jet lip line, at  $J=25$ . Both of these sets of samples are taken in the  $K=1$  plane and at all  $I$  indices from 60 to 141; that is, starting at a value of  $\eta/D$  of 0.5 and continuing downstream to the outflow boundary. As with the radial sampling, samples are recorded every fifteen time steps. The relative position of the segment of the jet lip line where predictions are computed is shown by the horizontal line in Figure 4.5.

Another parameter not investigated by either Yamamoto or Khavaran, Kresja, and Kim is the two point space-time cross correlation used by Chu [4] to compute the convection speed directly in a turbulent shear layer. This technique is described more fully in Section 4.7.3; for the purposes of the current discussion, it is sufficient to know that this technique requires values of  $u$  at two sampling points separated by a distance  $\xi$  and equidistant from a third point which is a distance  $\eta$  from the exit plane of the nozzle, as shown in Figure 4.4. For all the cross correlations done in this study, the center point between the two sampling points is set at a distance  $\eta/D$  of 8.86 ( $I=110$ ); this location was selected because it is roughly where fully developed jet flow begins, according to the results of Khavaran, Kresja, and Kim. Five pairs of points are then selected about this central point, as closely spaced as allowed by the grid; the separation distances and axial locations of these pairs of points are shown in Table 4.2. As with the axial turbulence intensity profiles, all samples are made in the  $K=1$  plane along the jet lip line ( $J=25$ ) and are recorded every fifteen time steps.

In all of the computations described here, a set of samples is taken at a regular interval of fifteen time steps. With an average time step size  $\Delta t$  of 0.67 microseconds,

| $\frac{\xi}{D}$ | $I_1$ | $I_2$ |
|-----------------|-------|-------|
| 0.000           | 110   | 110   |
| 0.662           | 109   | 111   |
| 1.325           | 108   | 112   |
| 1.990           | 107   | 113   |
| 2.659           | 106   | 114   |

Table 4.2: Locations and Spacing for Cross Correlation Sampling

this translates to a sampling frequency of 100 kHz. Based on the Nyquist frequency rule, this is sufficient to capture the upper ranges of the audible acoustic range (20 Hz to 20 kHz) accurately. However, sampling must also be done over a long enough time to cover the time scales of the lowest frequencies expected. For the simulations run for this study, predictions were computed over periods of thirty to fifty thousand iterations, or 20 to 33 milliseconds of simulated time. This translates to a lower bound of roughly 100 Hz (for thirty thousand iterations) to 60 Hz (for fifty thousand iterations) on the bandwidth of signals which can be captured accurately.

#### 4.7.2 Turbulence Calculations

A flow is generally described as turbulent if its behavior has strongly nonlinear and chaotic [13, 14, 20]. The characteristics of a turbulent flow are typically decomposed into a time average component and a perturbation component. In a steady state calculation, the influence of the perturbation components of the flow variables is typically handled using a turbulence model, such as the  $k$ - $\epsilon$  model used in the studies of Khavaran, Kresja, and Kim [11, 12]. However, in unsteady calculations such as

those used in this study, the perturbation components are captured implicitly as a part of the solution.

In this study, the main quantity of interest for turbulence analysis is the axial velocity component  $u$ , because it is the velocity component in the direction of the gross flow movement as well as the direction of the highest downstream noise levels [32]. At a given point in space and a given span of time, a time average velocity  $\bar{u}$  may be defined as

$$\bar{u}(x, y, z) = \frac{1}{t_1 - t_0} \int_{t_0}^{t_1} u(x, y, z, t) dt \quad (4.63)$$

A perturbation velocity  $u'$  may be defined as the difference of the instantaneous velocity  $u$  and the average speed  $\bar{u}$ :

$$u'(x, y, z, t) = u(x, y, z, t) - \bar{u}(x, y, z) \quad (4.64)$$

However, in a numerical simulation these quantities are not known in a continuous fashion in either time or space, and thus the integration used to compute the average velocity must be approximated using a Riemann sum, given by

$$\bar{u}_{i,j,k} = \frac{\sum_{n=1}^N u_{i,j,k}^n \Delta t_n}{\sum_{n=1}^N \Delta t_n} \quad (4.65)$$

Similarly, the perturbation velocity is defined in terms of the values of  $u$  and  $\bar{u}$  on the finite difference grid:

$$u'^n_{i,j,k} = u^n_{i,j,k} - \bar{u}_{i,j,k} \quad (4.66)$$

Another quantity of interest in the analysis of turbulence is the turbulence intensity. This is defined as the root mean square (RMS) value of the perturbation velocity [2]. In this study, it is normalized by the average speed on the center line:

$$I_{turb}|_{i,j,k} = \frac{\sqrt{(u')^2_{i,j,k}}}{\bar{u}_{i,l,k}} \quad (4.67)$$

### 4.7.3 The Convection Speed and Cross Correlation Coefficient

In flows with widely disparate mean velocities such as the ones examined in this study, it is difficult to select a single speed at which disturbances are convected downstream. This is particularly difficult in shear layers, where disparities in velocity are most evident. However, Papamoschou and Roshko introduced the concept of a mean convection speed in an interface between high speed and low speed flows with the same ratio of specific heats [7, 18]; the expression for this theoretical convection speed is

$$U_c = \frac{u_1 a_2 + u_2 a_1}{a_1 + a_2} \quad (4.68)$$

Here  $u$  refers to the mean velocity of the flow, and  $a$  refers to the mean sound speed; the subscript 1 indicates values within the jet core, while the subscript 2 indicates values in the surrounding flow. Papamoschou observed and Fourquette, Mungal, and Dibble confirmed that this formula under-predicts the actual speed at which disturbances are convected through shear layers for jet speeds at which compressibility effects may not be neglected.

Chu's experimental work [4] suggests an alternative method for the calculation of a convection speed. Chu computes a two point space-time cross correlation:

$$\mathcal{R}(\eta, \xi, \tau) = \frac{u(\eta + \frac{\xi}{2}, \tau)u'(\eta - \frac{\xi}{2})}{u^2(\eta)} \quad (4.69)$$

This correlation is a function of three parameters: the downstream distance  $\eta$  of the center point from the nozzle exit plane, the separation distance  $\xi$  between the two points correlated, and the delay time  $\tau$  between the samples at the correlated points. An further illustration of the notation used in this cross correlation is shown in Figure 4.4. For a set value of  $\eta$ , this results in a family of curves of  $\mathcal{R}$  versus  $\tau$  for specific values of  $\xi$ . On each curve, the location of the global maximum indicates the time  $\tau_*$  required for the largest number of disturbances to propagate from the upstream point at  $\eta - \frac{\xi}{2}$  to the downstream point at  $\eta + \frac{\xi}{2}$ . The convection speed can be then computed as the ratio of the separation distance  $\xi$  to the delay time  $\tau_*$ , as shown in Equation 4.70. Moreover, the envelope of all curves for a given  $\eta$  should result in the same convection speeds.

$$U_c = \frac{\xi}{\tau_*} \quad (4.70)$$

For working purposes, it is convenient to normalize  $\xi$  by the nozzle exit diameter and the delay time by the diameter and jet exit velocity. This results in the following normalization:

$$\frac{U_c}{U_j} = \frac{\left(\frac{\xi}{D}\right)}{\left(\frac{U_j \tau_*}{D}\right)} \quad (4.71)$$

This equation uses the time  $\tau_*$  required for a disturbance to be convected across a distance  $\xi$  to compute the convection speed. This may be related back to the

theoretical prediction for convection speed in Equation 4.68 by considering the one dimensional gas dynamics example of a disturbance in a piston with a gas moving with speed  $u_1$  inside it. A disturbance in the piston will create two pressure waves, one moving to the right at  $u_1 + a_1$  and one moving to the left at  $u_1 - a_1$ , where  $a_1$  is the sound speed inside the piston. If the outer wall of the piston is then replaced with a massless constant pressure interface to a second flow moving with speed  $u_2$ , it follows that a disturbance in the second flow must create pressure waves moving at  $u_2 + a_2$  to the right and  $u_2 - a_2$  to the left. Furthermore, at the interface between the two flows, the speed at which pressure waves from a disturbance travel must be a function of the flow speeds ( $u_1, u_2$ ) and sound speeds ( $a_1, a_2$ ) in both flows.

#### 4.7.4 Scaling Lengths

Of the many scaling parameters which could be used to characterized the viscous behavior of a jet, the one used in this study to characterize the growth of the shear layer is the momentum thickness  $\theta$  [13, 14, 20]. The momentum thickness is a length scale associated with the momentum deficit caused by the mixing between the jet core and the surrounding flow. Using the jet speed  $U_j$  as the normalizing speed, the momentum thickness may be defined as

$$\theta = \int_0^\infty \frac{\bar{u}}{U_j} \left(1 - \frac{\bar{u}}{U_j}\right) d\tau \quad (4.72)$$

Numerically, this expression may be computed using a trapezoidal rule approximation:

$$\theta = \frac{1}{2} \sum_{i=2}^N \left[ \frac{\bar{u}_i}{U_j} \left(1 - \frac{\bar{u}_i}{U_j}\right) + \frac{\bar{u}_{i-1}}{U_j} \left(1 - \frac{\bar{u}_{i-1}}{U_j}\right) \right] [\tau_i - \tau_{i-1}] \quad (4.73)$$

Following the analysis for a turbulent circular jet described by Schlichting [20], the streamwise velocity  $\bar{u}$  is

$$\bar{u} = \frac{3}{8\pi} \frac{K}{\epsilon_0 \pi} \frac{1}{\left(1 + \frac{\eta^2}{4}\right)^2} \quad (4.74)$$

Here  $\eta$  is a similarity parameter:

$$\eta = \frac{1}{4\epsilon_0} \sqrt{\frac{3K}{\pi}} \frac{r}{x} \quad (4.75)$$

$K$  refers to the constant kinematic momentum of the jet:

$$K = 2\pi \int_0^\infty u^2 r dr = \frac{1}{4} \pi D^2 U_j^2 \quad (4.76)$$

$\epsilon_0$  is the virtual turbulent kinematic viscosity. Schlichting [20] shows that this must be constant in the case of a circular jet and gives the following empirical relation (based on the experimental findings of Reichardt):

$$\epsilon_0 = 0.0161K \quad (4.77)$$

Substituting Equations 4.74 through 4.77 in Equation 4.72 and carrying out the integration with respect to  $r$  results in the following relation:

$$\frac{\theta}{D} = \frac{x}{D} D \left( 0.2165 - \frac{0.3193}{U_j} \right) \quad (4.78)$$

However, as  $U_j$  is known to be on the order of 2400 fps. and thus much larger than 0.3193, the last term in this expression may be neglected. This results in

$$\frac{\theta}{D} = 0.2165 D \left( \frac{x}{D} \right) \quad (4.79)$$

Another scaling parameter useful in characterizing the spreading of the jet is the half speed radius  $r_{1/2}$ . This is the radius at which the average speed  $\bar{u}$  at a given axial location has slowed to half the value of the jet speed  $U_j$ . This length may be estimated from plots of the radial velocity profiles or computed by interpolating between the recorded values in the sampling files.

## CHAPTER 5

### NUMERICAL RESULTS

This chapter describes the results of the numerical simulations performed in this study. General characteristics of the numerical simulations are described. Visualizations of the entropy and shock function fields are described. Velocity and turbulence predictions are discussed and compared with results from axisymmetric calculations and experiment. The two point space-time correlation is used to compute the mean convection speed along the jet shear layer.

#### 5.1 General Behavior

A total of six simulation cases were run for this investigation. There were two parameters varied in these cases: the ratio of specific heats  $\gamma$  and the ratio of static pressures between the nozzle inlet and the surrounding freestream flow. In addition, in the first four cases a supersonic outflow condition was used, while in the fifth and sixth cases a subsonic out flow condition was imposed. Other characteristic flow parameters, such as the Reynolds number based on diameter  $Re_D$ , the maximum jet speed  $U_j$ , and the external flow speed  $U_{ext}$ , were determined from results of the numerical simulation. The conditions used in each of these cases are summarized in Table 5.1.

| Run # | $\gamma$ | $Re_D$  | $p_{\infty,1}$<br>$\frac{lb.}{ft^2}$ | $p_{\infty,2}$<br>$\frac{lb.}{ft^2}$ | $U_j$<br>$\frac{ft}{s}$ | $U_{ext}$<br>$\frac{ft}{s}$ | Outflow<br>B.C. | Iterations<br>of Sampling |
|-------|----------|---------|--------------------------------------|--------------------------------------|-------------------------|-----------------------------|-----------------|---------------------------|
| 1     | 1.400    | 592,004 | 6646.                                | 2407.                                | 2558.77                 | 391.43                      | Supersonic      | 45k–65k                   |
| 2     | 1.353    | 567,511 | 6646.                                | 2407.                                | 2459.00                 | 461.84                      | Supersonic      | 45k–85k                   |
| 3     | 1.353    | 496,636 | 5816.                                | 2117.                                | 2327.20                 | 449.90                      | Supersonic      | 45k–75k                   |
| 4     | 1.400    | 505,899 | 5816.                                | 2117.                                | 2383.42                 | 396.77                      | Supersonic      | 55k–85k                   |
| 5     | 1.353    | 567,511 | 5816.                                | 2117.                                | 2462.90                 | 426.60                      | Subsonic        | N/A                       |
| 6     | 1.353    | 496,636 | 5816.                                | 2407.                                | 2288.66                 | 429.66                      | Subsonic        | 50k–75k                   |

Table 5.1: Flow Conditions for Simulation Cases

The formulation and implementation of the outflow boundary condition had a strong effect on the character of the resulting flow. The subsonic outflow condition used in runs 5 and 6 did not perform as expected, and in fact drove the simulation to a non-physical solution after approximately seventy thousand iterations. Because of this, the results from runs 5 and 6 are not considered in this discussion; however, as the formulation and implementation of a subsonic outflow boundary condition for this configuration remains a topic of ongoing research, these runs provided valuable experience. The supersonic outflow condition, while not having the stability problems associated with the subsonic outflow condition described above, also had its share of problems. This condition caused the jet core flow to remain supersonic over the entire length of the computational domain. As a result, the mixing of the jet with the surrounding flow was retarded, and only a small amount of jet spreading was observed.

Variations in the inflow static pressures also had a strong effect on the general character of the flow. The main influence of varying the inlet pressures was to change

the jet speed and external flow speed, which in turn modified the strength and orientation of the oblique shock wave downstream of the nozzle exit plane. Variation in the specific heat ratio  $\gamma$  appeared to have only minimal influence on the character of the flow.

## 5.2 Flow Visualizations

Visualization is generally used to give the analyst a qualitative feel for the behavior of a flow simulation. However, visualization can also be used to extract quantitative information under certain circumstances which would be difficult to find otherwise. In this study, visualization was used not only for qualitative examination of pressure, density, Mach number, and jet spreading rate, but also for quantitative location of oblique shocks within the jet core flow. As it is difficult to see details in gray-scale plots of density, pressure, or Mach number due to the lack of contrast, this document will concern itself with visualizations of entropy and the shock function.

### 5.2.1 Entropy and Jet Spreading Behavior

The rate at which the jet core spreads and mixes with the surrounding flow is of particular interest in several fields, including acoustic analysis. The spreading of the jet can be shown by examining the entropy field. The expected result is a strong increase in entropy through the shear layer between the jet core and the surrounding flow, which then drops off progressively as the jet core mixes out.

Figures 5.1 through 5.4 show the instantaneous entropy fields at the end of the last iteration of each respective simulation. In these figures, the exterior flow appears as an extremely dark, almost black region. The jet core is a dark gray, although lighter than the exterior flow. The shear layer can be seen as the thin light gray regions

separating the exterior flow from the jet core. From these figures, several qualitative observations can be made.

In run 1 (Figure 5.1), the jet spreads slightly over a distance of two nozzle diameters downstream of the exit plane. From there, the jet begins to contract out to a distance of about twenty-five diameters, whereupon it begins spreading again. This contraction is not seen in theoretical or experimental results, and it appears to be caused by the supersonic outflow boundary condition keeping the jet core supersonic over the length of the computational domain. The shear layer appears to remain the same thickness over the entire domain. The overall entropy field ranges from about 67,080 in the exterior flow to 80,510 at the trailing edge of the nozzle lip.

In run 2 (Figure 5.2), similar behavior is observed. The jet spreads slightly for the first two nozzle diameters downstream of the nozzle exit, then begin to contract. However, in this case there is no spreading observed near the downstream boundary, as was seen in run 1; this is due to the higher external flow speed in run 2 than in run 1. The shear layer appears to be somewhat thicker than in run 1, but again remains a constant thickness. The overall range of entropy values of entropy in run 2 are higher than in run 1, from 76,810 in the exterior flow to 88,870 at the nozzle lip.

Run 3 (Figure 5.3) exhibits spreading behavior more similar to run 1 than run 2. The pattern of spreading over the first two nozzle diameters followed by contraction is again observed. As in run 2, no spreading is observed near the downstream boundary, again due to a higher external flow speed than in run 1. However, the shear layer in run 3 appears thinner than that in run 2 and is comparable in thickness to that in run 1. The range of entropy values is also similar to those observed in run 2, from 77,300 in the exterior flow to 88,650 at the nozzle trailing edge.

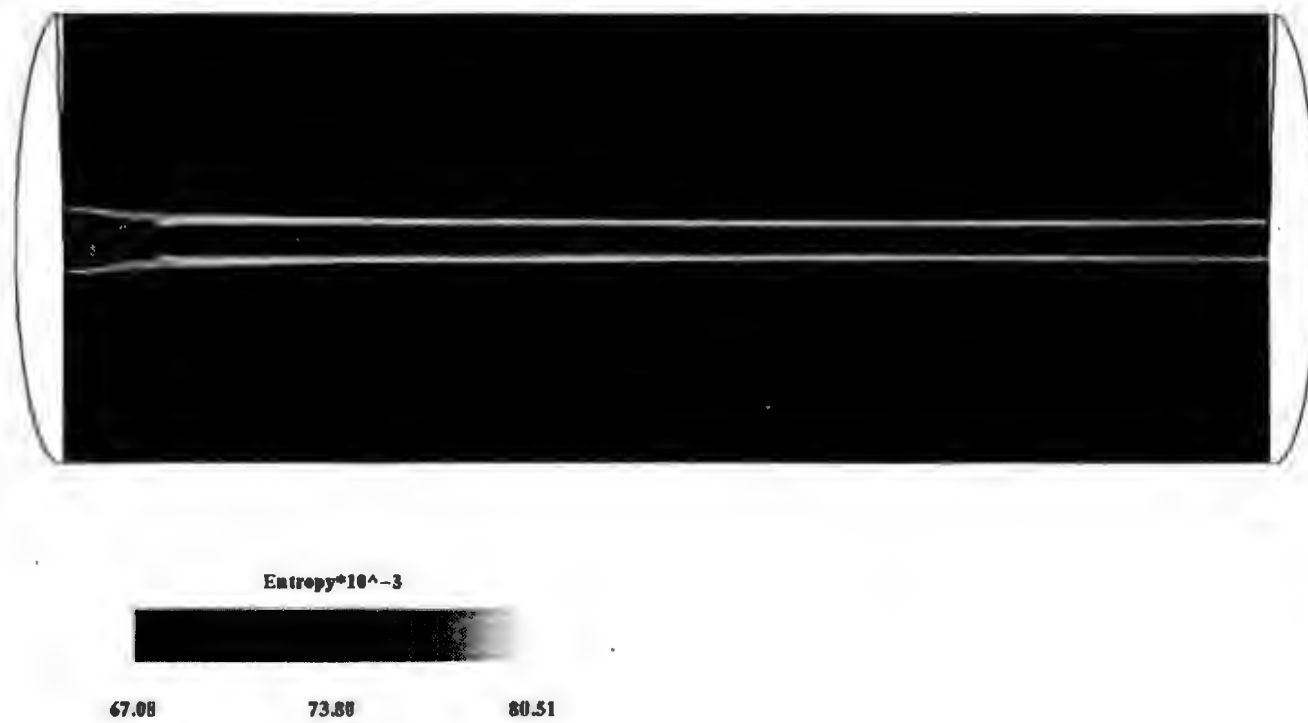


Figure 5.1: Entropy for Run 1 at 43.2 ms. (65000 Iterations)

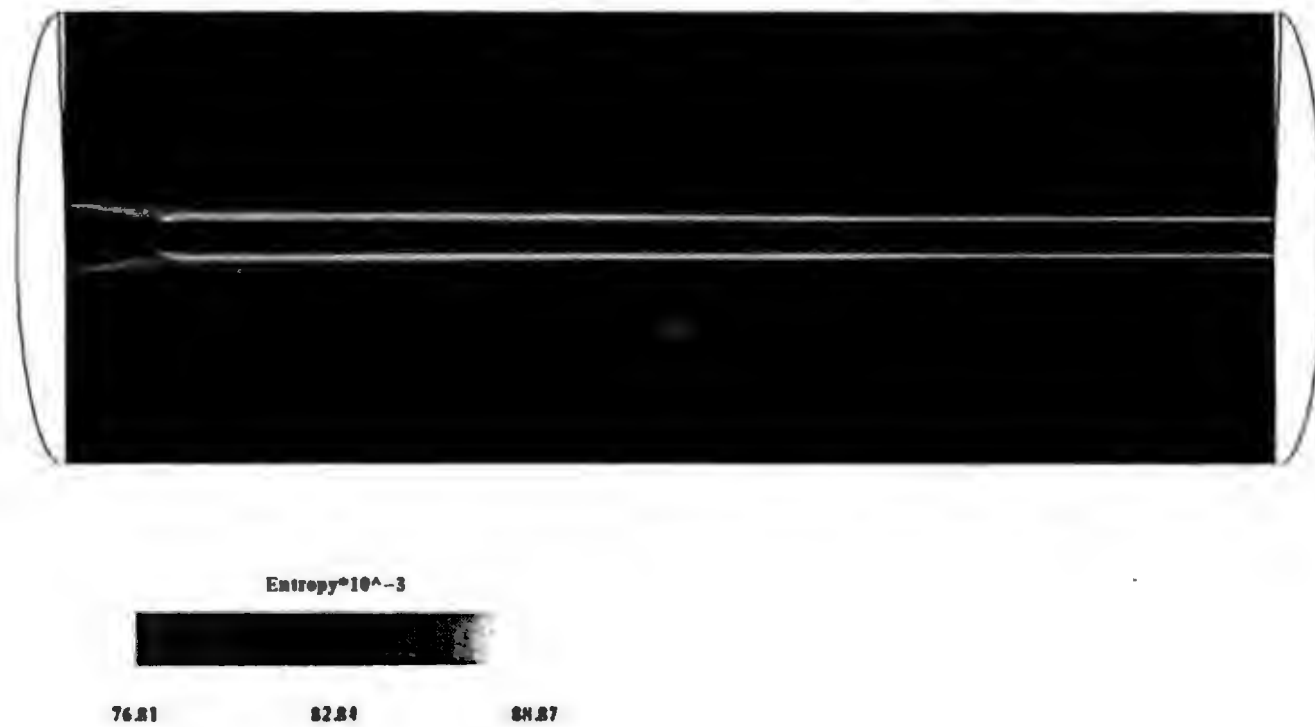


Figure 5.2: Entropy for Run 2 at 57.8 ms. (85000 Iterations)

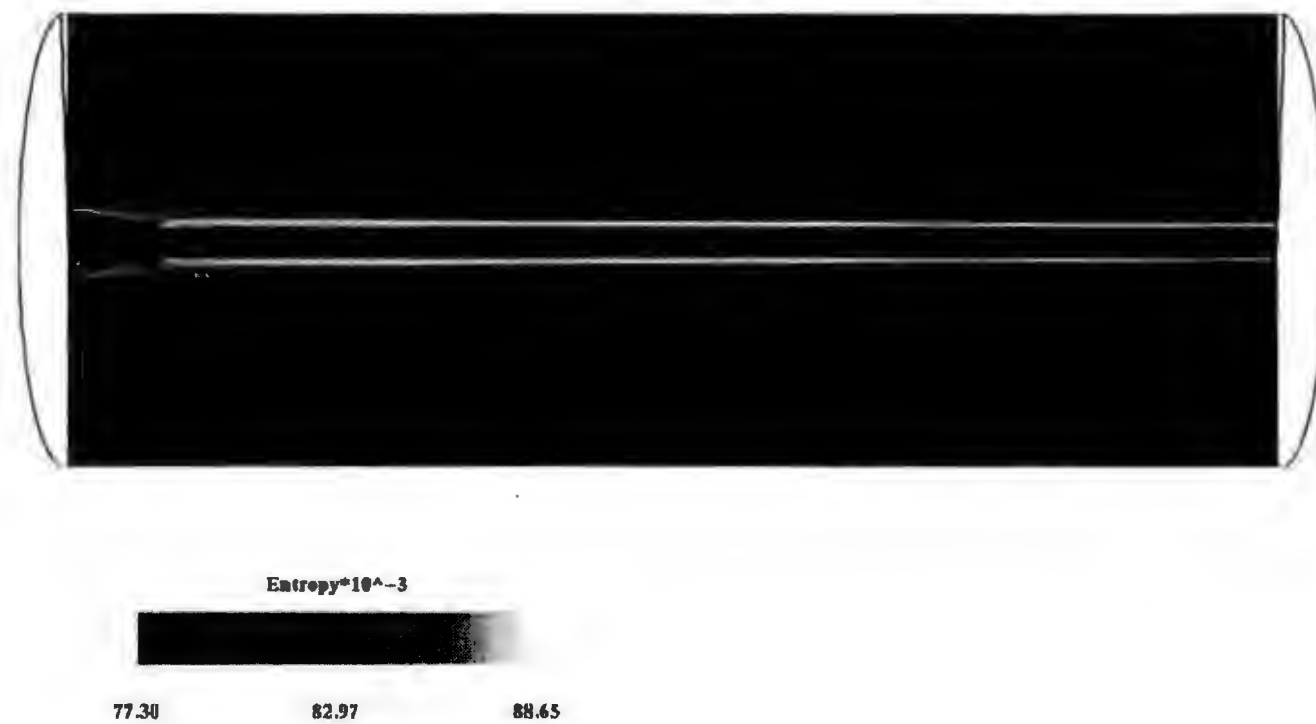


Figure 5.3: Entropy for Run 3 at 52.0 ms. (75000 Iterations)



Figure 5.4: Entropy for Run 4 at 58.3 ms. (85000 Iterations)

The spreading behavior observed in run 4 is markedly different from that observed in the other cases. The jet continues to spread much further downstream than in the other cases, reaching its maximum width between eight and nine exit diameters downstream of the exit plane. The jet then begins to contract, and continues to do so over the rest of the computational domain. This increased spreading over the previous runs is caused by a combination of a lower jet speed and a smaller difference between the jet and external flow speeds than in runs 1 and 2. The shear layer is significantly thicker than those observed in runs 1 and 3 and even approximately 25% thicker than that of run 2. The entropy field range is roughly comparable to that in run 1, from 69,900 in the exterior flow to 80,820 at the nozzle trailing edge.

These figures consistently show that, in cases with a supersonic outflow condition, the jet core remains highly coherent and spreads very little from the nozzle exit plane to the outflow plane. This is consistent with the findings of Papam schou and Roshko; Chuech, Lai and Faeth; and Fourquette, Mungal, and Dibble [18, 5, 7], all of which indicate that compressibility effects tend to retard the jet spreading rate. Another factor which affects the mixing rate is the lip trailing edge geometry. In these simulations, the nozzle lip's trailing edge is treated as a blunt termination, and there are only two grid lines defining the upper and lower surfaces of the nozzle wall without any grid lines in between to account for the finite radius at the trailing edge.

### 5.2.2 Shock Locations

As discussed in Section 4.6, the shock function is useful for the identification and location of shock waves in a compressible flow. Visualizations of the instantaneous shock function at the end of each case are illustrated in Figures 5.5 through 5.8. In

these figures, areas in which shocks are present are light gray or white, while regions of accelerating flow are dark gray or black. These visualizations consistently show a first oblique shock roughly three fourths of an exit diameter downstream of the nozzle exit plane, and most of them also show a secondary compression roughly two and one-half exit diameters downstream of the nozzle exit plane. The locations and angles (here noted as  $\beta$ ) of these are summarized in Table 5.2.

In run 1 (Figure 5.5), the oblique shock intersects the center line at a distance of 0.808 exit diameters from the nozzle exit plane. The angle of the oblique shock with the center line is  $39.9^\circ$ . The secondary compression intersects the center line at 2.604 exit diameters from the nozzle exit and at an angle of  $31.1^\circ$ . A rippling effect can be seen in the secondary compression and in the jet core downstream of the secondary compression. This is a result of the non-physical reflection of pressure waves from the center line caused by the first order differences used in the center line boundary condition for this case. These reflections were further evidenced by oscillations in lines of constant pressure and Mach number near the center line, as mentioned in the discussion of the center line boundary condition in Section 4.5.1.

The shock behavior in run 2 (Figure 5.6) is very similar to that of run 1 near the nozzle; the locations and angles for both the oblique shock and the secondary compression are identical as those observed in run 1. As the only difference between the input parameters of runs 1 and 2 is the ratio of specific heat  $\gamma$ , this is not unexpected. However, the secondary compression in run 2 appears to be stronger and more coherent than in run 1; this is probably a result of the fact that run 2 was run for significantly longer than run 1. Furthermore, the rippling caused by pressure wave reflections from the center line is less prevalent in the secondary compression

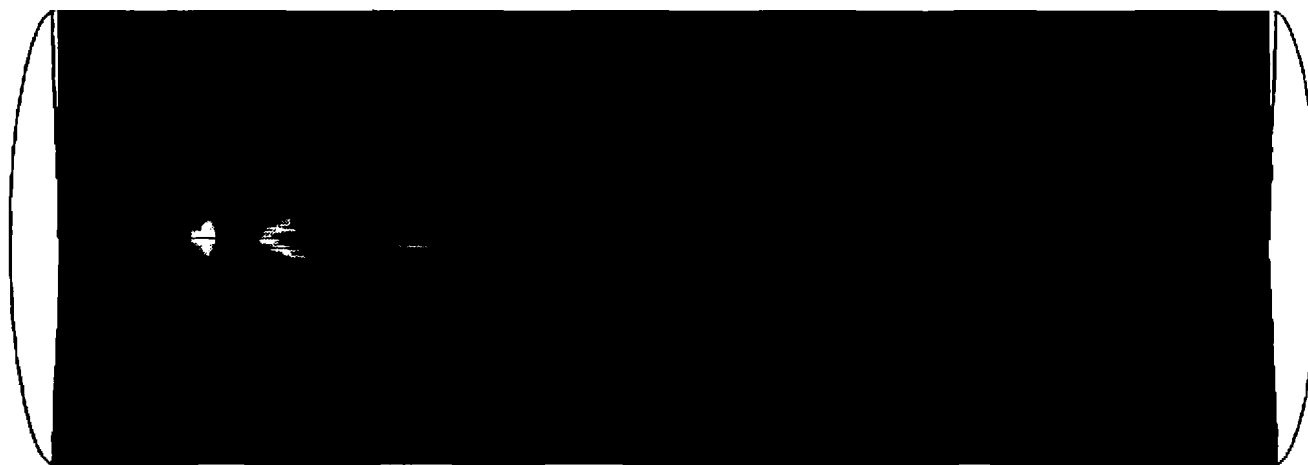


Figure 5.5: Shock Function for Run 1 at 43.2 ms. (65000 Iterations)



Figure 5.6: Shock Function for Run 2 at 57.8 ms. (85000 Iterations)

in run 2 than in run 1. However, the influence of the center line boundary condition in run 2 extends into the exterior flow, as evidenced by the longitudinal striations in the shock function near the middle of the computational domain in the lighter gray area outside the very dark gray of the jet core.

In run 3 (Figure 5.7), the overall shock structure is closer to the nozzle exit plane than in runs 1 and 2 as a result of the lower inlet pressure. It should be noted that this may be difficult to see in the figure due to size constraints. The oblique shock intersects the center line at a distance of 0.731 nozzle exit diameters from the exit plane and at an angle of  $40.6^\circ$ . The secondary compression intersects the centerline at 2.479 exit diameters and at an angle of  $37.1^\circ$ ; it appears to be stronger than the secondary compression in run 1 but weaker than that of run 2. The rippling caused by the center line boundary condition has a similar influence on the secondary compression to that observed in run 2. The effects of the center line boundary condition on the exterior flow are much weaker than that observed in run 2, as shown by a reduction in the longitudinal striations outside the jet core.

The shock behavior in run 4 (Figure 5.8) is characterized by a much stronger influence of pressure reflections from the center line boundary condition than in the previous cases. The oblique shock appears to be significantly less coherent than in the previous runs due to the rippling effects from the center line boundary condition, as can be seen from its smeared appearance in the figure; it intersects the center line at a distance of 0.731 exit diameters from the nozzle exit plane and at an angle of  $36.5^\circ$ . This similarity to run 3 is expected, as runs 3 and 4 differ only in the ratio of specific heats  $\gamma$ . However, a coherent secondary compression is not observed in this case; it appears to have been dissipated by the reflections from the center line. The

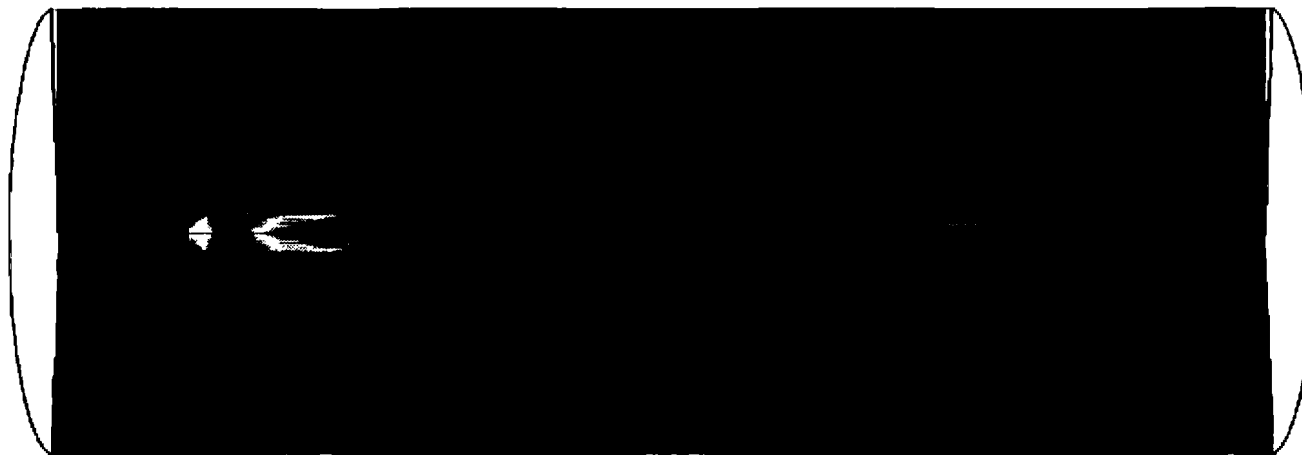


Figure 5.7: Shock Function for Run 3 at 52.0 ms. (75000 Iterations)



Figure 5.8: Shock Function for Run 4 at 58.3 ms. (85000 Iterations)

| Run # | Shock Wave |         | 2nd Compression |         |
|-------|------------|---------|-----------------|---------|
|       | $\eta/D$   | $\beta$ | $\eta/D$        | $\beta$ |
| 1     | 0.808      | 39.9°   | 2.604           | 31.1°   |
| 2     | 0.808      | 39.9°   | 2.604           | 31.1°   |
| 3     | 0.731      | 40.6°   | 2.479           | 37.1°   |
| 4     | 0.731      | 36.5°   | N/A             | N/A     |

Table 5.2: Locations and Angles of Shock Wave and Secondary Compression

influence of the center line boundary condition on the exterior flow appears to be weaker than in run 2 and comparable to run 3, as evidenced by the reduced amount of rippling in the shock function near the outflow boundary.

### 5.3 Turbulence Predictions

Visualization, while a powerful tool for analysis, does not tell the whole tale of the physics of the jet flow. Visualization is useful for examining the entire computational domain for patterns and structure. However, for a more detailed analysis, working with the entire domain is unwieldy and expensive in terms of storage requirements; instead, values are extracted from selected regions of the computational domain for examination. With carefully chosen locations, this approach allows for direct comparison between the predictions of a numerical simulation and the findings of an experiment.

Four quantities representative of the flow's turbulent behavior are discussed here. First, time averages of the axial velocity are shown as a function of radius for several axial locations. Second, the axial turbulence intensity is also shown as a function of radius at the same axial locations as the axial velocity. Third, the axial turbulence

intensity along the jet lip line is shown as a function of axial location. Finally, a two point space-time cross correlation is computed to determine the characteristic delay time for velocity disturbances passing through a specified axial location on the jet lip line. For each of these quantities, the numerical solution was sampled at a frequency of 100 kHz; the iteration ranges over which each case solution was sampled are shown in in Table 5.1. To insure that the unsteady effects observed were due to unsteadiness in the governing equation and not numerical effects, the calculations were run for one characteristic time (the time required for a fluid particle to travel from the inflow boundary to the outflow boundary in the exterior flow) before starting the sampling procedure.

### 5.3.1 Radial Distribution of Axial Velocity

Figures 5.9, 5.10, 5.12, and 5.13 illustrate the radial profiles of the time average axial velocity in the K=1 grid plane at five axial locations:  $\eta/D = 0.20, 0.96, 2.24, 4.21$ , and  $8.86$ . In each figure, the time average velocity is normalized by the maximum jet speed  $U_j$  for that case. These profiles confirm the overall lack of spreading shown in the entropy plots from Section 5.2.1.

In run 1 (Figure 5.9), the flow continues to accelerate from the nozzle exit plane to the oblique shock, indicated by the higher center line speed at  $\eta/D = 0.96$  (after the oblique shock) than at  $\eta/D = 0.20$  (before the oblique shock). The flow then decelerates slows over the rest of the computational domain, as evidenced by the small drop in center line speed from  $\eta/D = 2.24$  to  $\eta/D = 8.86$ . The spreading behavior observed in Figure 5.1 is confirmed by these velocity profiles; the radial location at which the axial speed has dropped to 50% of the jet speed  $U_j$  moves from

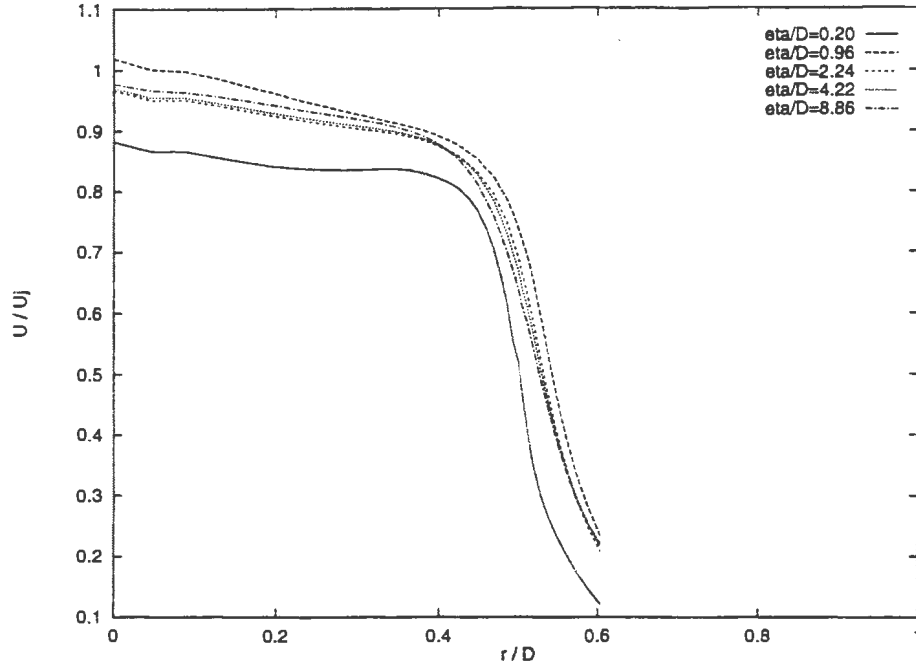


Figure 5.9: Radial Velocity Profiles from Run 1

$\tau_{1/2}/D = 0.501$  at  $\eta/D = 0.20$  to  $0.542$  at  $\eta/D = 0.96$ , then contracts to  $0.532$  at  $\eta/D = 2.24$  and slowly decreases further at  $\eta/D = 4.21$  and  $\eta/D = 8.86$ . This contraction is not what one would expect to occur physically, and it appears to be caused by the supersonic outflow condition retarding the mixing of the jet core.

In run 2 (Figure 5.10), the flow again accelerates from the nozzle exit to the oblique shock. However, in this case the flow does not decelerate downstream of the shock system, as shown by the constant center line speeds on the  $\eta/D = 2.24$ ,  $\eta/D = 4.21$ , and  $\eta/D = 8.86$  curves. This, like the lack of spreading observed in the entropy plots in Section 5.2.1, appears to be a result of the supersonic outflow condition inhibiting the mixing of the jet core. The thicker shear layer observed in the entropy plot shown in Figure 5.2 is also shown by these curves;  $\tau_{1/2}/D$  is  $0.407$  at  $\eta/D = 0.20$ ,  $0.460$

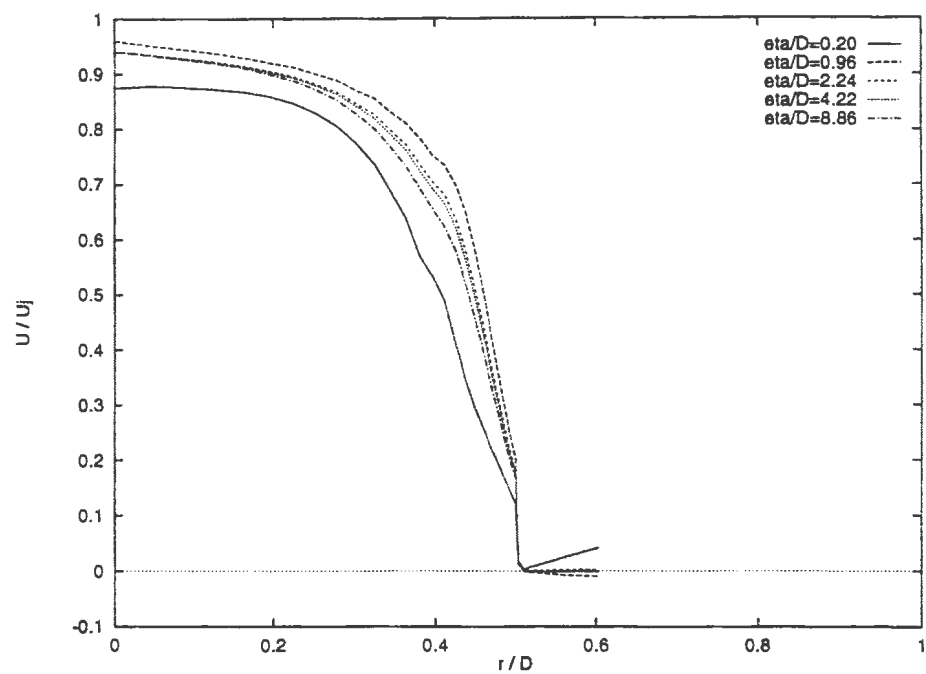


Figure 5.10: Radial Velocity Profiles from Run 2

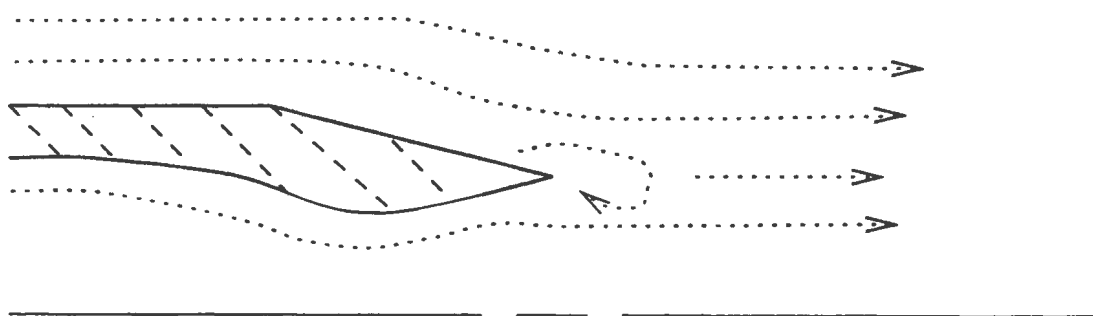


Figure 5.11: Streamlines Near Nozzle Separation Region

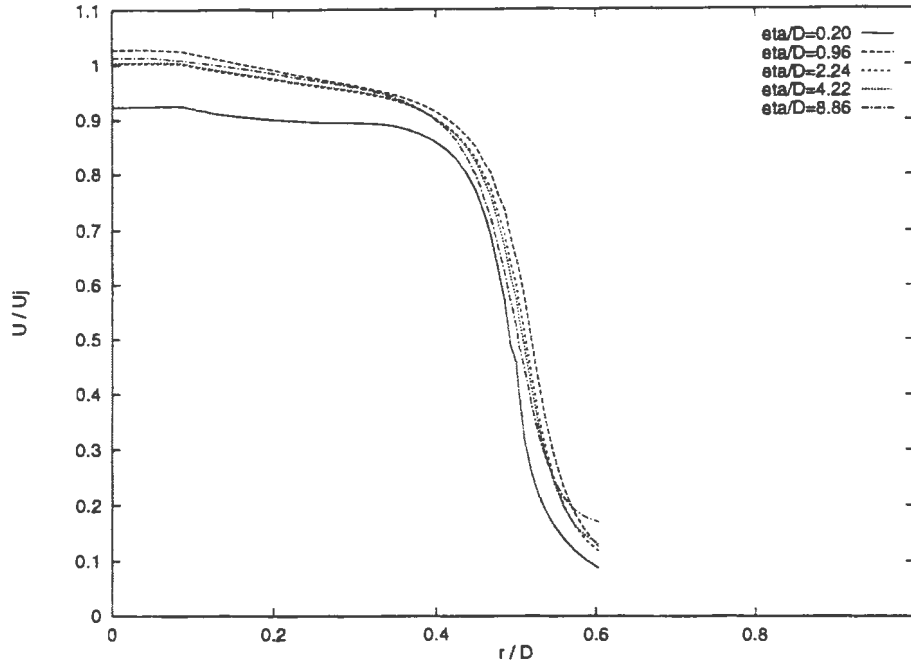


Figure 5.12: Radial Velocity Profiles from Run 3

at  $\eta/D = 0.96$ , and 0.450 at  $\eta/D = 2.24$ . This increase in the shear layer thickness is caused by a separation region on the back slope of the nozzle; a sketch of the streamlines near the separation region is shown in Figure 5.11, although their size and spacing is exaggerated for emphasis. This separation region is further indicated by the drop of all velocity profiles to zero at  $r/D = 0.50$  in this case and the negative velocities between  $r/D = 0.50$  and  $r/D = 0.60$  seen in the  $\eta/D = 0.96$  curve.

Run 3 (Figure 5.12) exhibits behavior very similar to that observed in run 1. The jet core again accelerates from the nozzle exit plane to the oblique shock, then decelerates slightly over the rest of the domain. The thin shear layer and spreading pattern observed in Figure 5.3 is confirmed here;  $r_{1/2}/D$  is 0.492 at  $\eta/D = 0.20$ , 0.519

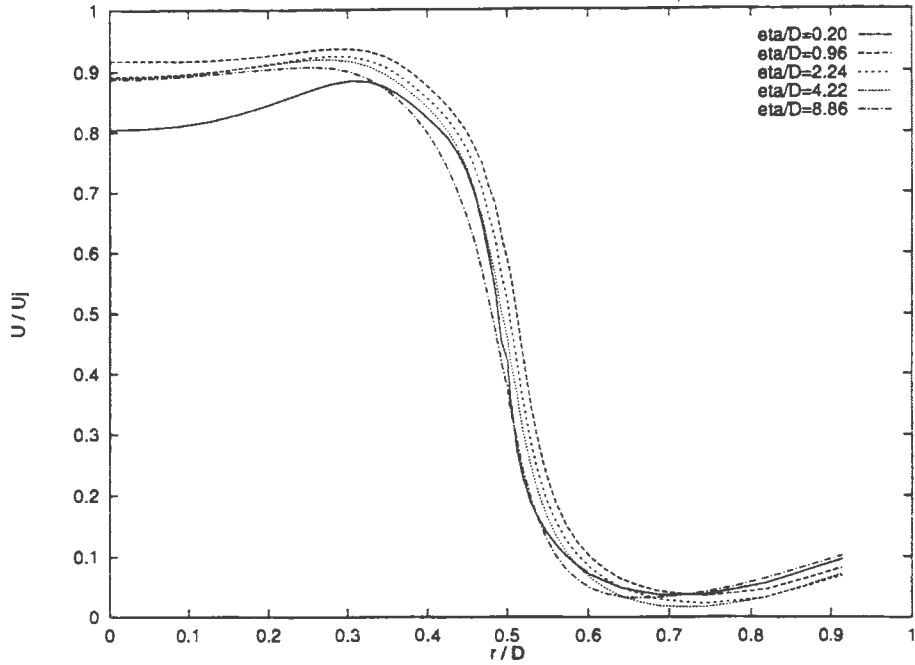


Figure 5.13: Radial Velocity Profiles from Run 4

at  $\eta/D = 0.96$ , and 0.511 at  $\eta/D = 2.24$ . The region of zero or negative velocity observed in run 2 is not observed in run 3.

Run 4 (Figure 5.13) again shows significantly different behavior from that observed in the other cases. In run 4, the maximum speed in a given profile is not observed at the center line; instead, the maximum speed is near  $r/D = 0.30$ . This appears to be caused by a transient phenomenon related to the outflow boundary condition; similar although much more severe behavior was observed in runs 5 and 6 before they became unstable. As with runs 1 and 3, the jet core accelerates to the oblique shock then decelerates slightly over the rest of the domain. However, a separation region similar to that observed in run 2 is evidenced by the near zero velocities near  $r/D$

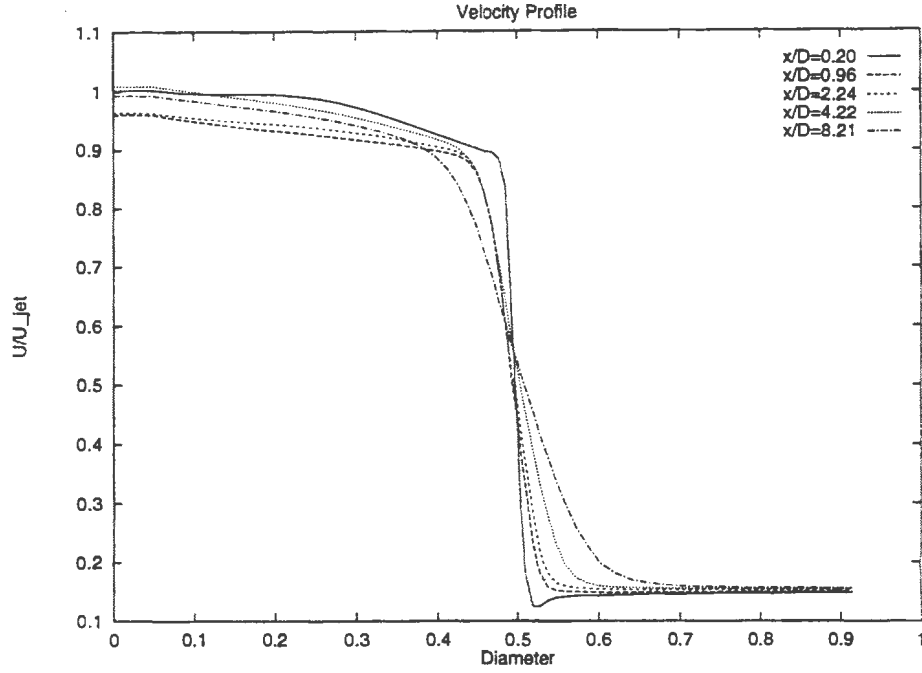


Figure 5.14: Radial Velocity Profiles from Preliminary Axisymmetric Calculations by Scott [26]

$= 0.70$ ; this separation is not as strong as that observed in run 2, as no negative velocities are observed.

By way of comparison, the velocity profiles shown in Figure 5.14 from preliminary results for an axisymmetric calculation on the same geometry performed by Scott [26] show somewhat different behavior from the 3D solutions computed for this study. The jet core at  $\eta/D = 0.20$  is much more sharply defined in the axisymmetric results than in any of the cases from this study; this probably can be attributed to the increased dissipation in the 3D form of the governing equations over that in the axisymmetric form of the equations. However, the axisymmetric solution shows significantly more

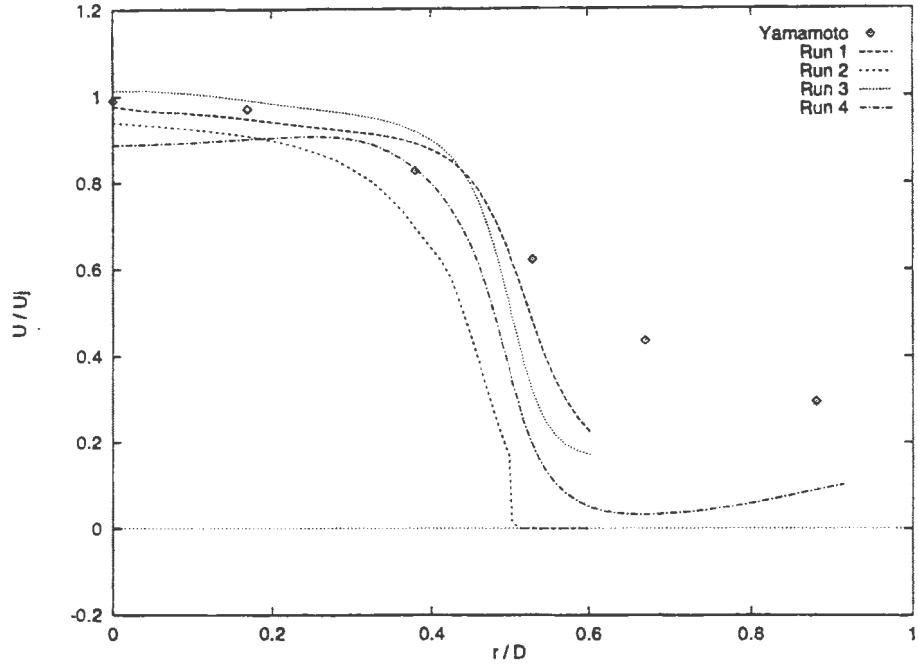


Figure 5.15: Comparison of Radial Velocity Profiles at  $\eta/D = 8.86$  with Experimental Data from Yamamoto, et al. [32]

spreading than any of the 3D solutions, as evidenced by the flattening of the profile with distance downstream.

Further comparisons may be made with laser velocimetry data from the experiment study of this nozzle done by Yamamoto and colleagues [32]. Figure 5.15 shows the radial velocity profiles from runs 1 through 4 at  $\eta/D = 8.86$  in comparison to laser velocimetry data taken at  $\eta/D = 8.7$ . As with the axisymmetric results described above, the experimental results show significantly more spreading than the 3D numerical solutions. The experimental results also do not indicate the presence of the separation region observed in runs 2 and 4.

|            |            |            |            |            |            |
|------------|------------|------------|------------|------------|------------|
| $\eta/D =$ | 0.20       | 0.96       | 2.24       | 4.21       | 8.86       |
| Run #      | $\theta/D$ | $\theta/D$ | $\theta/D$ | $\theta/D$ | $\theta/D$ |
| 1          | 0.089      | 0.052      | 0.066      | 0.065      | 0.062      |
| 2          | 0.080      | 0.056      | 0.064      | 0.064      | 0.066      |
| 3          | 0.068      | 0.037      | 0.045      | 0.045      | 0.044      |
| 4          | 0.102      | 0.079      | 0.081      | 0.080      | 0.089      |
| Theory     | 0.019      | 0.093      | 0.218      | 0.410      | 0.862      |

Table 5.3: Summary of Momentum Thicknesses Results

Using the trapezoidal rule approximation given in Equation 4.73, the momentum thickness was computed from these velocity profiles for each run and axial location. The results of these calculations are summarized in Table 5.3, along with theoretical predictions using the relation for  $\theta/D$  given by Equation 4.79 as derived in Section 4.7.4. These results confirm some of the qualitative observations made in Section 5.2.1 with respect to shear layer thickness; the momentum thickness in run 4 is consistently the largest of the four, and the momentum thickness in run 3 is consistently the smallest. The momentum thicknesses in runs 1 and 2 are very similar in both magnitude and behavior with respect to downstream distance, which is to be expected as the two runs differ only in the value of  $\gamma$  used. The computed results for momentum thickness differ greatly from those predicted by the theory discussed in Section 4.7.4, and there are several reasons for this. The theoretical prediction, which states that the momentum thickness is proportional to the distance downstream, does not take account compressibility in the expression used for the virtual turbulent kinematic viscosity, and compressibility effects cannot be neglected in this flow. Furthermore, the supersonic outflow boundary condition appears to significantly retard the mixing

| $\eta/D =$ | 0.20        | 0.96        | 2.24        | 4.21        | 8.86        |
|------------|-------------|-------------|-------------|-------------|-------------|
| Run #      | $r_{1/2}/D$ | $r_{1/2}/D$ | $r_{1/2}/D$ | $r_{1/2}/D$ | $r_{1/2}/D$ |
| 1          | 0.501       | 0.542       | 0.531       | 0.529       | 0.526       |
| 2          | 0.407       | 0.460       | 0.450       | 0.448       | 0.440       |
| 3          | 0.492       | 0.519       | 0.511       | 0.509       | 0.502       |
| 4          | 0.488       | 0.512       | 0.502       | 0.493       | 0.481       |

Table 5.4: Summary of  $U = 50\%U_j$  Radial Location Results

of the jet core, thus causing the momentum thickness not to grow in the expected linear manner.

Table 5.4 summarizes the  $r_{1/2}/D$  results from runs 1 through 4. These confirm the qualitative observations made in Section 5.2.1 that the jet spreads in the first two exit diameters downstream of the exit plane and then contracts over the rest of the domain. This disagrees with the theoretical discussion of turbulent circular jets by Schlichting [20], which predicts that the the jet width should increase linearly with distance downstream. As with the momentum thickness described above, this discrepancy has several causes. First, Schlichting's prediction neglects compressibility, which is a strong influence on this flow. Also, the apparent retardation of mixing caused by the supersonic outflow boundary condition causes the jet width no to grow in the predicted linear fashion.

### 5.3.2 Radial Distribution of Axial Turbulence Intensity

Figures 5.16 through 5.19 show radial profiles of the turbulence intensity in the  $K=1$  grid plane at five axial locations:  $\eta/D = 0.20, 0.96, 2.24, 4.21$ , and  $8.86$ . As

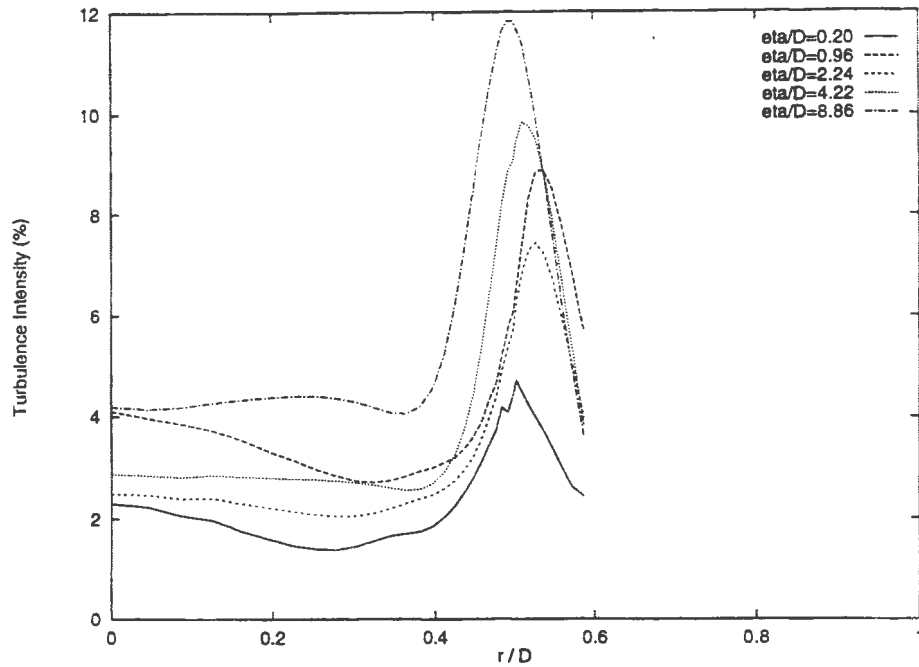


Figure 5.16: Radial Turbulence Intensity Profiles from Run 1

with the velocity profiles, the turbulence intensity is normalized by the maximum jet speed  $U_j$  and is expressed as a percentage.

In run 1 (Figure 5.16), the turbulence intensity at the center line increases with distance downstream, from 2.29% at  $\eta/D = 0.20$  to 4.19% at  $\eta/D = 8.86$ . This increase indicates that there is some mixing occurring within the jet core. Each of the turbulence intensity curves remain relatively flat from the center line to  $r/D = 0.4$ . Approaching the shear layer, the turbulence intensity decreases, then increases dramatically upon entering the shear layer. The locations of the peaks in turbulence intensity match closely with the  $r_{1/2}/D$  values shown in Table 5.4. The magnitude of the peaks increase with distance downstream, from 4.70% at  $\eta/D = 0.20$  to 11.82% at  $\eta/D = 8.86$ . After the peak, the turbulence intensity again decreases rapidly. The

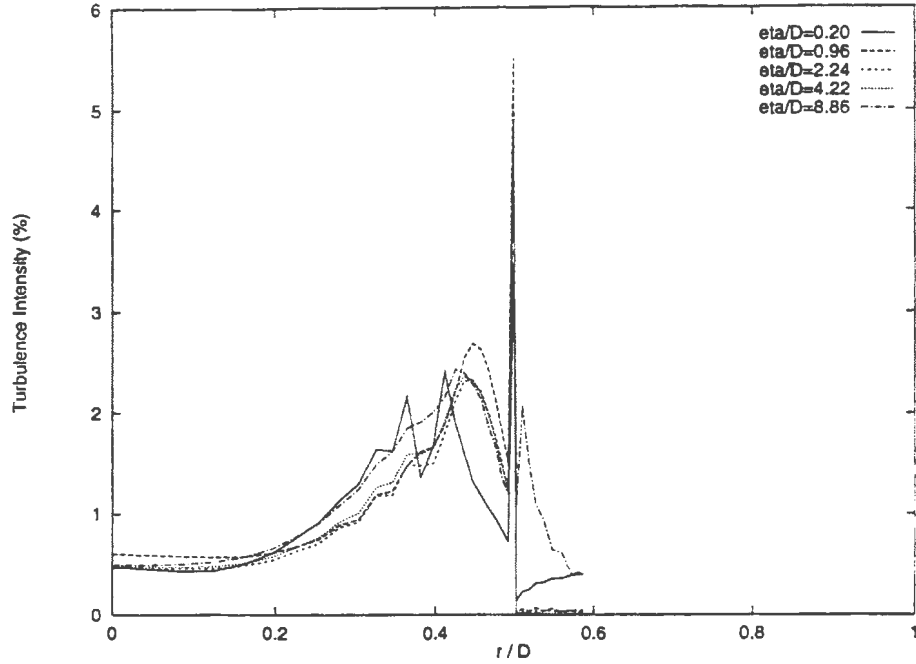


Figure 5.17: Radial Turbulence Intensity Profiles from Run 2

half-width of each peak corresponds roughly to the momentum thickness  $\theta/D$  values shown in Table 5.3.

Run 2 (Figure 5.17) shows significantly different behavior, due to the separation region described in Section 5.3.1. The turbulence intensity at the center line remains almost constant at 0.50%, which indicates that the mixing of the jet core does not increase with distance downstream. The curves then remain flat to  $r/D = 0.2$ , where they begin to increase in a manner similar to that observed in run 1. As in run 1, the turbulent intensity curves peak near the  $r_{1/2}/D$  values listed in Table 5.4; these peaks range from 2.41% at  $\eta/D = 0.20$  to 2.69% at  $\eta/D = 0.96$ . However, at the nozzle lip line ( $r/D = 0.50$ ), the turbulence intensity increases sharply, ranging from 3.46% at  $\eta/D = 0.20$  to 5.50% at  $\eta/D = 0.96$ ; this appears to be related to the

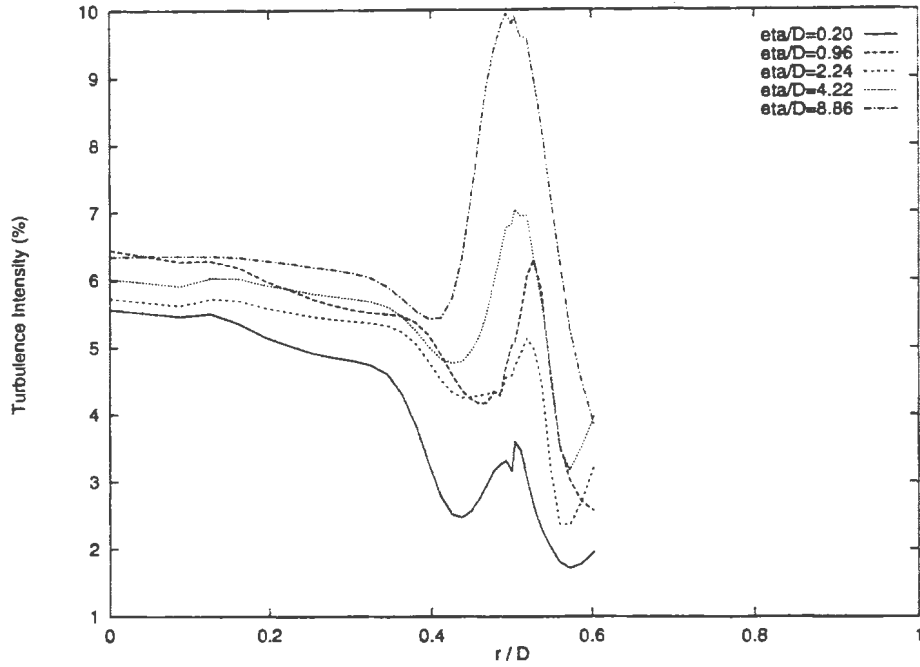


Figure 5.18: Radial Turbulence Intensity Profiles from Run 3

separation region observed in Section 5.3.1. After this sharp increase at the lip line, the turbulence intensity drops immediately to approximately zero.

In contrast to run 2, run 3 (Figure 5.18) exhibits turbulence intensity behavior very similar to that in run 1. The turbulence at the center line again increases with distance downstream, ranging from 5.56% at  $\eta/D = 0.20$  to 6.34% at  $\eta/D = 8.86$ , indicating that there is some degree of mixing within the jet core. The profiles then decrease near  $r/D = 0.4$ ; this decrease is large enough at  $\eta/D = 0.20$  and  $\eta/D = 2.24$  that the peak turbulence intensity in the shear layer is actually lower than that at the center line. This decrease in turbulence intensity appears to be related to the inflow pressures, as it appears in the two runs with lowered inflow pressures (5816 psf in runs 3 and 4) but not in the runs with higher inflow pressures (6646 psf in runs 1

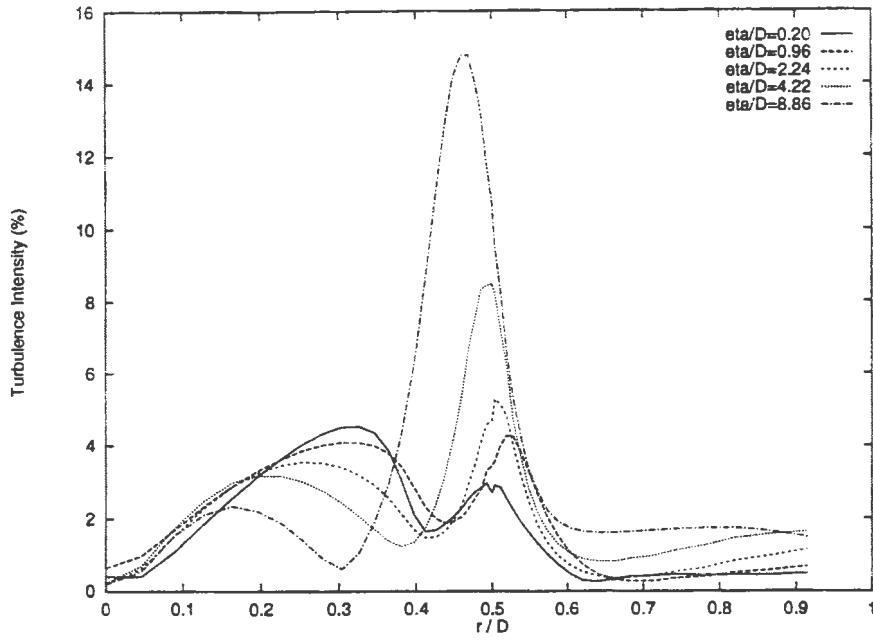


Figure 5.19: Radial Turbulence Intensity Profiles from Run 4

and 2). The peak turbulence intensity in the shear layer again increases with distance downstream, ranging from 3.58% at  $\eta/D = 0.20$  to 9.95% at  $\eta/D = 8.86$ . Once again, the radial locations of the peak turbulence intensity in the shear layer correspond to the  $r_{1/2}/D$  locations listed in Table 5.4, and the half-width of the peaks correspond roughly to the  $\theta/D$  values shown in Table 5.3.

Run 4 (Figure 5.19) shows characteristics similar to both run 2 and run 3. As in run 2, the turbulence intensity at the center line remains relatively constant with distance downstream at 0.50%, indicating that the mixing within the jet core does not increase with distance downstream. However, here the turbulence intensity begins to increase almost immediately; this is due to the fact that the velocity near  $r/D = 0.3$  is larger than that at the center line due to a transient phenomenon related to the

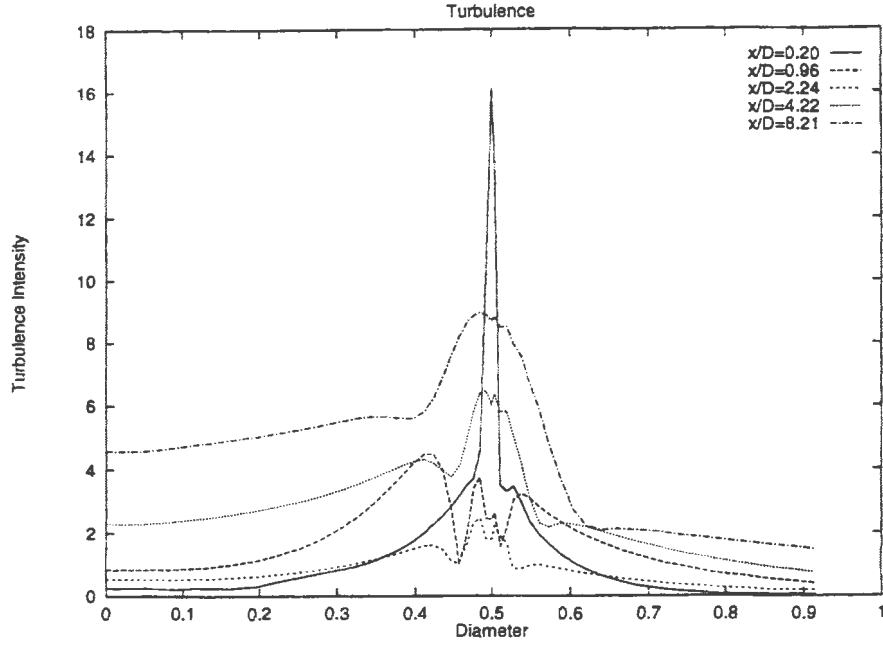


Figure 5.20: Radial Turbulence Intensity Profiles from Preliminary Axisymmetric Calculations by Scott [26]

outflow boundary condition, as discussed in Section 5.3.1. The drop in turbulence intensity near  $r/D = 0.4$  is such that the peak turbulence intensity in the shear layer at  $\eta/D = 0.20$  is lower than that in the jet core, similar to run 3; again, this appears to be related to the inflow pressure. As in runs 1 and 3, the peak turbulence intensity in the shear layer increases with distance downstream, ranging from 2.97% at  $\eta/D = 0.20$  to 14.77% at  $\eta/D = 8.86$ . The radial locations of the shear layer peaks in turbulence intensity again correspond to the  $\tau_{1/2}/D$  locations in Table 5.4.

By comparison, the radial turbulence intensity profiles from preliminary axisymmetric calculations by Scott [26] in Figure 5.20 show behavior similar in many respects to that observed in runs 1 and 3. The turbulence intensity at the center line increases with distance downstream as in runs 1 and 3, ranging from about 0.5% at  $\eta/D = 0.20$

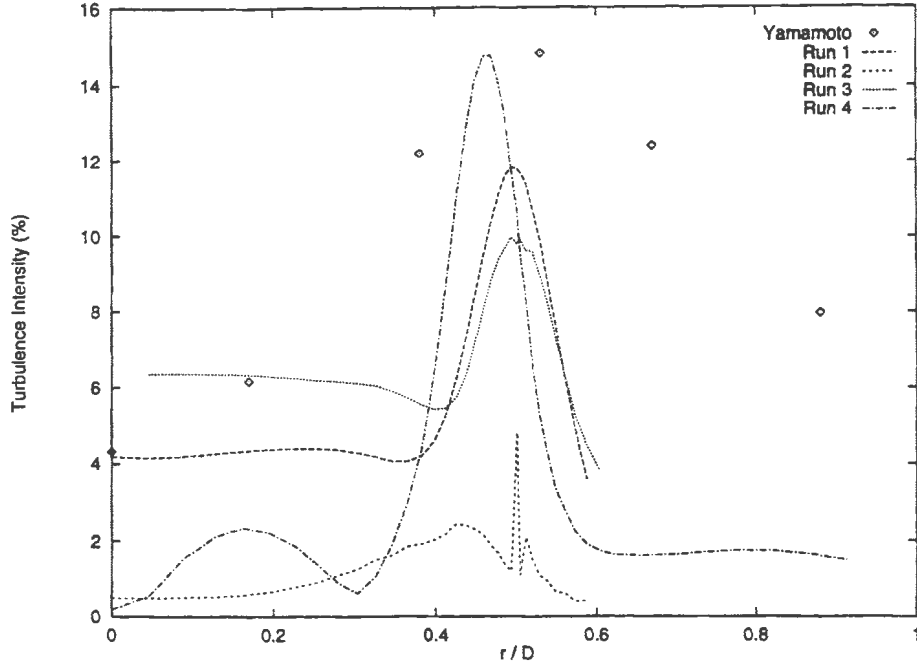


Figure 5.21: Comparison of Radial Turbulence Intensity Profiles at  $\eta/D = 8.86$  with Experimental Data from Yamamoto, et al. [32]

to 4.5% at  $\eta/D = 8.86$ . However, the peak turbulence intensity in the shear layer does not strictly increase with distance downstream as it did in runs 1 and 3; instead, it decreases from a value of 16% at  $\eta/D = 0.20$  to 2.4% at  $\eta/D = 2.24$ , then increases to 8.8% at  $\eta/D = 8.86$ . The locations of these peaks remain nearly constant at  $r/D = 0.5$ , but the width of the peaks grows with downstream distance as predicted by the theoretical momentum thickness expression shown in Equation 4.79.

Further comparisons may be made with the laser velocimetry data from the experimental study of Yamamoto, et al [32]. Figure 5.21 compares the computed radial turbulence intensity profiles from runs 1 through 4 at  $\eta/D = 8.86$  to laser velocimetry data for turbulence intensity taken at  $\eta/D = 8.7$ . As in the velocity profile results

discussed in Section 5.3.1, the experimental results show significantly more spreading than the 3D numerical results, as evidenced by the much larger width of the peak in turbulence intensity in the shear layer. However, the experimental turbulence intensities near the center line compare favorably in magnitude with those computed in runs 1 and 3, and the magnitude of the peak in the shear layer is similar to those computed for runs 2, 3 and 4.

### 5.3.3 Axial Distribution of Turbulence Intensity

Figures 5.22 through 5.25 illustrate the axial turbulence distributions along the lip line for runs 1 through 4, respectively. All these profiles show a sharp dip in the turbulence intensity at  $\eta/D = 2.126$ . This dip is caused by the intersection of the oblique shock with the shear layer, as can be seen by careful comparison of the entropy plots shown in Figures 5.1 through 5.4 with their corresponding shock function plots in Figures 5.5 through 5.8. Again, this may be difficult to see in the printed figures due to size constraints. In runs 1 (Figure 5.22) and 3 (Figure 5.24), the turbulence intensity increases to a maximum value near  $\eta/D = 16$ , then decreases to the outflow boundary. This is the behavior of a fully developed jet, as similar behavior is observed in the results of Yamamoto, et al. [32], and it confirms the observations made in Section 5.3.2 that there is a degree of mixing present in the jet cores in runs 1 and 3.

In runs 2 (Figure 5.23) and 4 (Figure 5.25), the turbulence intensity increases essentially linearly from the dip at  $\eta/D = 2.126$  to the outflow boundary. This behavior is a result of the separation on the nozzle exterior observed in these two runs as described in Section 5.3.1. This continued increase in turbulence intensity along

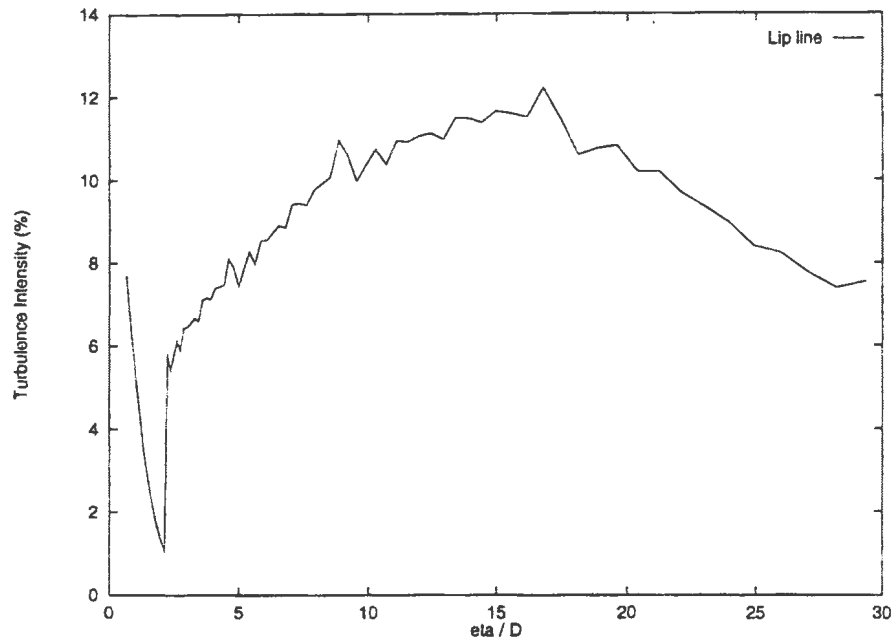


Figure 5.22: Lip Line Axial Turbulence Intensity Profile from Run 1

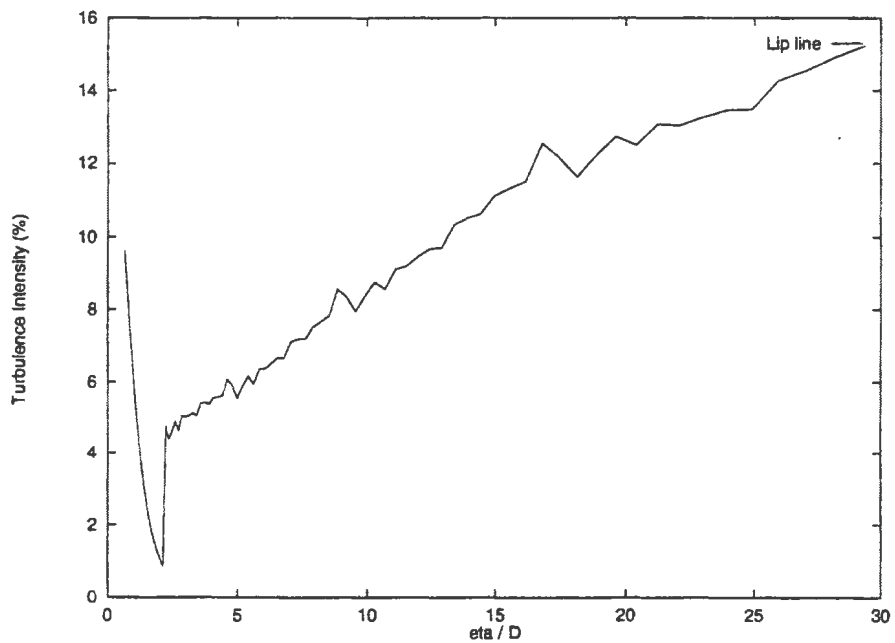


Figure 5.23: Lip Line Axial Turbulence Intensity Profile from Run 2

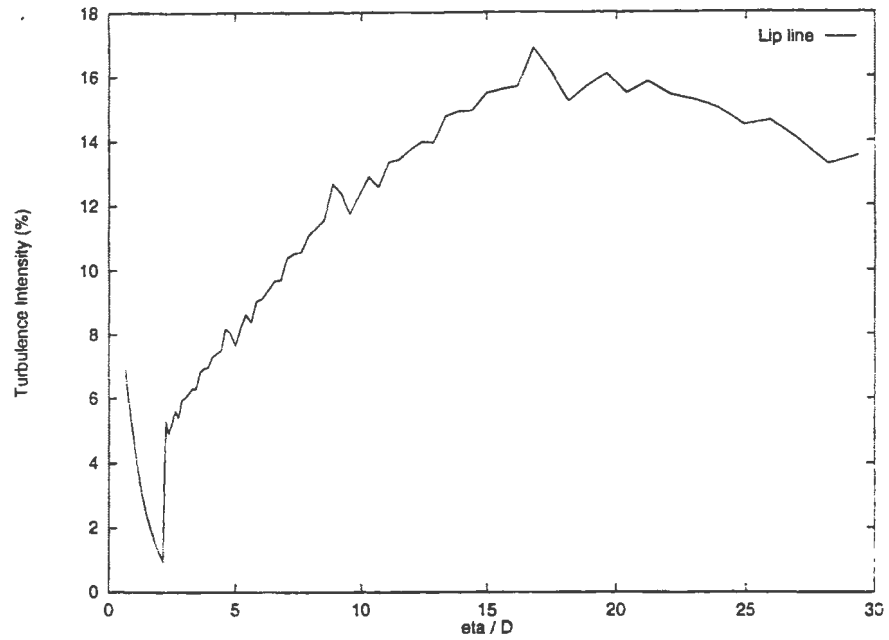


Figure 5.24: Lip Line Axial Turbulence Intensity Profile from Run 3

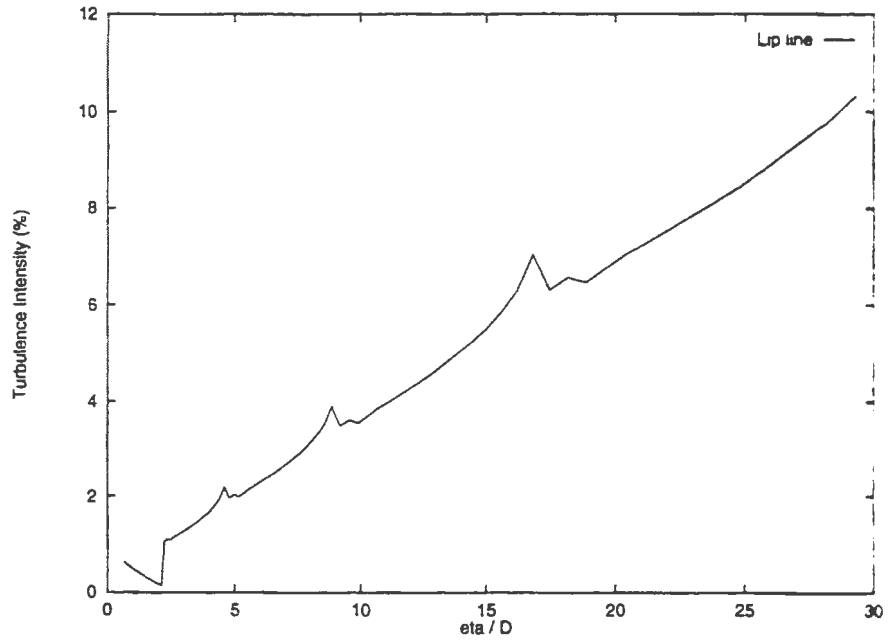


Figure 5.25: Lip Line Axial Turbulence Intensity Profile from Run 4

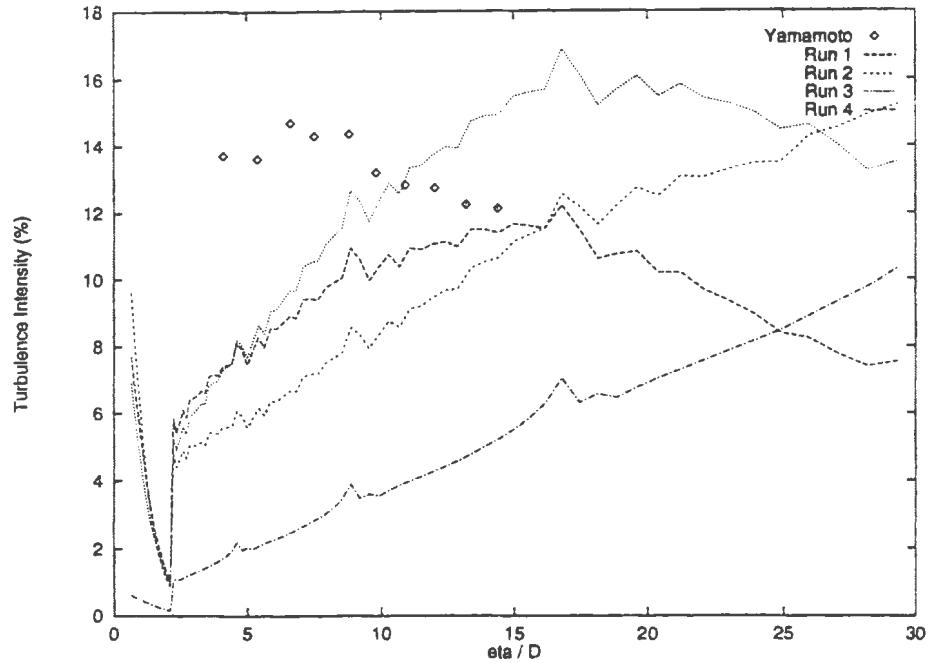


Figure 5.26: Comparison of Lip Line Axial Turbulence Intensity Profiles with Experimental Data from Yamamoto, et al. [32]

the lip line indicates that the jet core still has not fully mixed with the surrounding flow before reaching the outflow boundary. This confirms the observations made in Section 5.3.2 that little mixing takes place in the jet cores in runs 2 and 4.

Figure 5.26 compares the axial turbulence intensity profiles computed in runs 1 through 4 to laser velocimetry data from the experimental study of Yamamoto, et al [32]. The computed turbulence intensity profiles are stretched considerably in the axial direction relative to the experimental results; this is due to the lack of spreading observed in the numerical results. However, the shape of the turbulence profiles observed in runs 1 and 3 are quite similar to that seen in the experimental results but stretched in the axial direction; the profiles from runs 1 and 3 show the same trend of increase, plateau, and gradual decrease as is seen in the experimental

data. The computed profiles from runs 2 and 4 do not compare as well with the experimental results, as the jet core in these runs have not fully mixed with the surrounding flow.

#### 5.3.4 Cross Correlation and Convection Speed Predictions

Figures 5.27 through 5.30 show the two point space-time cross correlation given by Equation 4.69 for runs 1 through 4, respectively. In each of these figures, the cross correlation is plotted for five separation distances:  $\xi/D = 0.000, 0.662, 1.325, 1.990$ , and  $2.659$ . The  $\tau U_j/D$  value at the global maximum of each curve is used to compute a convection speed; the results of these calculations are shown in Table 5.5, along with theoretical predictions using Papamoschou and Roshko's formula for convection speed [18]; given by Equation 4.68. This formula is a weighted average of the core and exterior flow speeds where the weighting factors are related to the sound speeds in the core and exterior flows.

In run 1 (Figure 5.27), the correlation results in a family of similar curves which are shifted to the right as the separation distance  $\xi/D$  increases. This shift to the right indicates the increased time required for a disturbance to travel from the upstream sampling point to the downstream sampling point as the separation distance is increased. The  $\tau U_j/D$  locations of the global maxima of these curves are used to compute the normalized convection speed  $U_c/U_j$ , ranging from 0.393 at  $\xi/D = 0.662$  to 0.401 at  $\xi/D = 1.989$ . The normalized convection speed predicted by theory is 0.468; this is somewhat surprising, as the theoretical prediction of Papamoschou and Roshko (Equation 4.68) is known to under-predict the convection speed where compressibility may not be neglected.

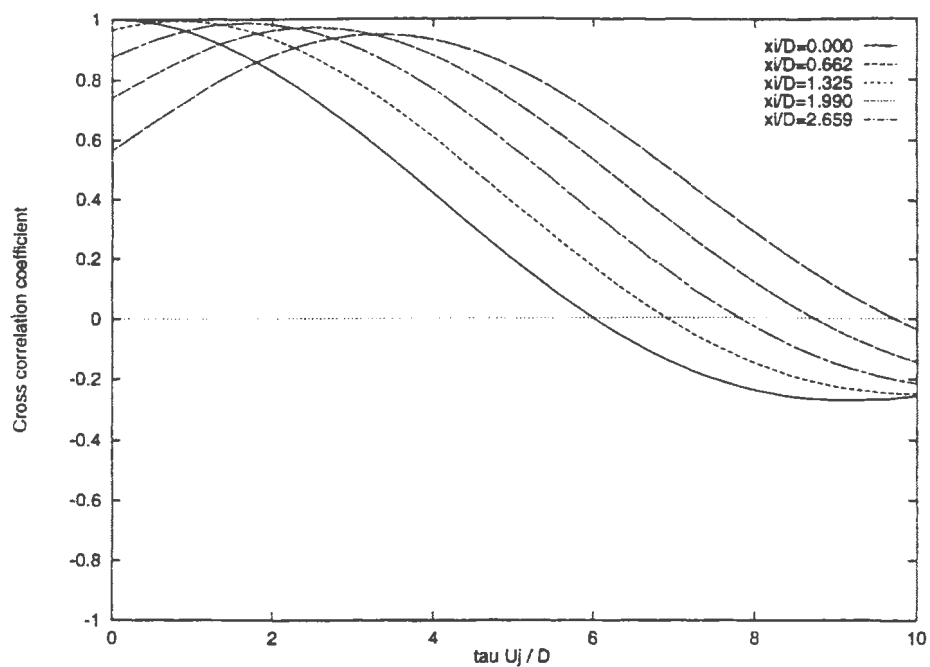


Figure 5.27: Cross Correlation for Run 1

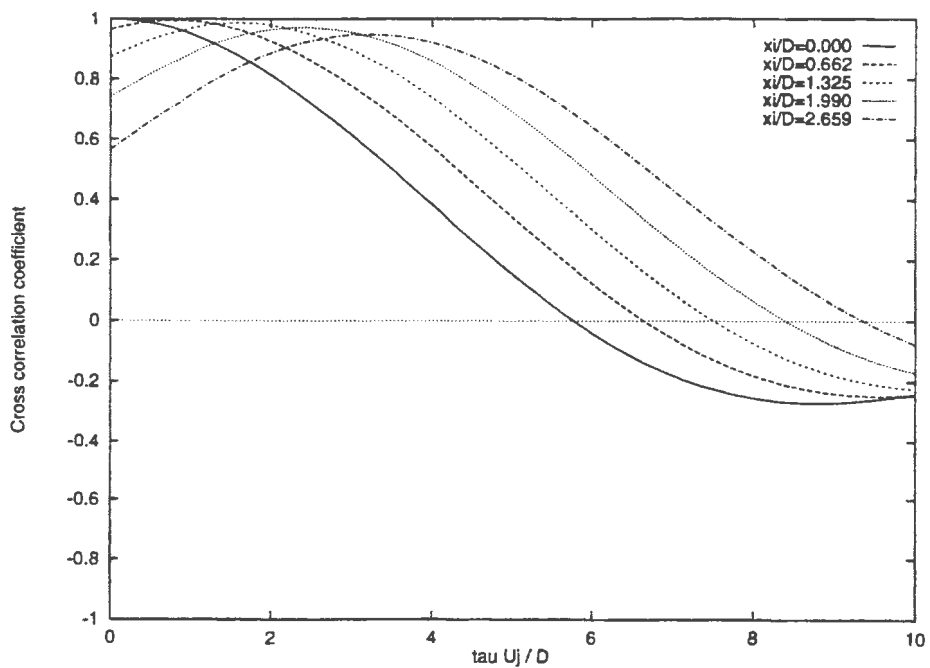


Figure 5.28: Cross Correlation for Run 2

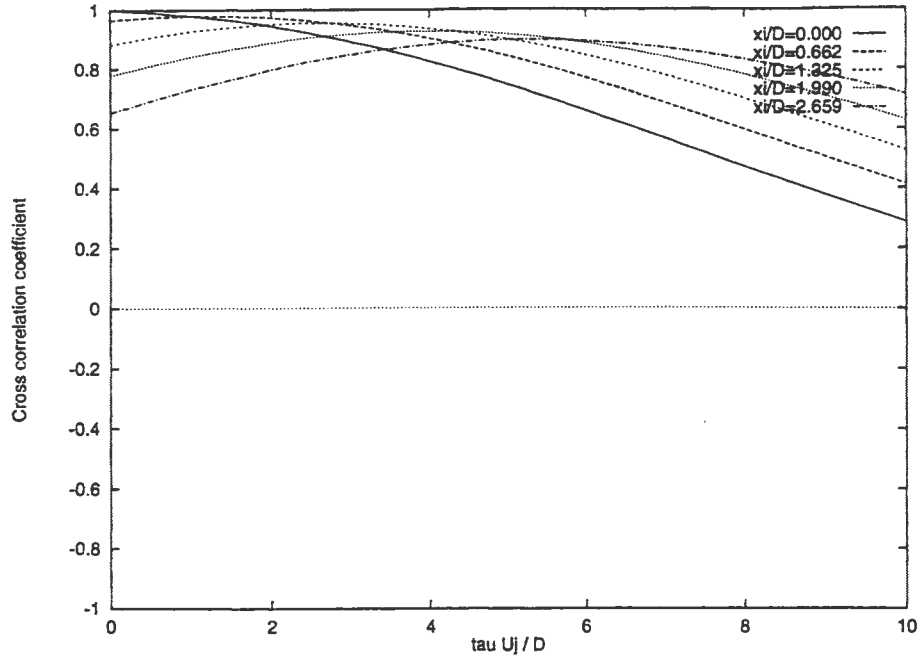


Figure 5.29: Cross Correlation for Run 3

The correlations from run 2 (Figure 5.28) result in a family of curves very similar to those observed in run 1. The curves are again shifted to the right as  $\xi/D$  is increased. The computed normalized convection speeds range from 0.511 at  $\xi/D = 1.324$  to 0.526 at  $\xi/D = 2.657$ . The normalized convection speed predicted by theory is 0.479; given the known tendency of the theory to under-predict with compressibility, this is in agreement with the computed results.

In run 3, the correlation curves are stretched more along the  $\tau U_j/D$  axis, although the curves are again shifted to the right as  $\xi/D$  is increased. The computed values for normalized convection speed range from 0.508 at  $\xi/D = 2.657$  to 0.520 at  $\xi/D = 0.662$ . The normalized convection speed predicted by theory is 0.492, which again agrees well with the computed results.

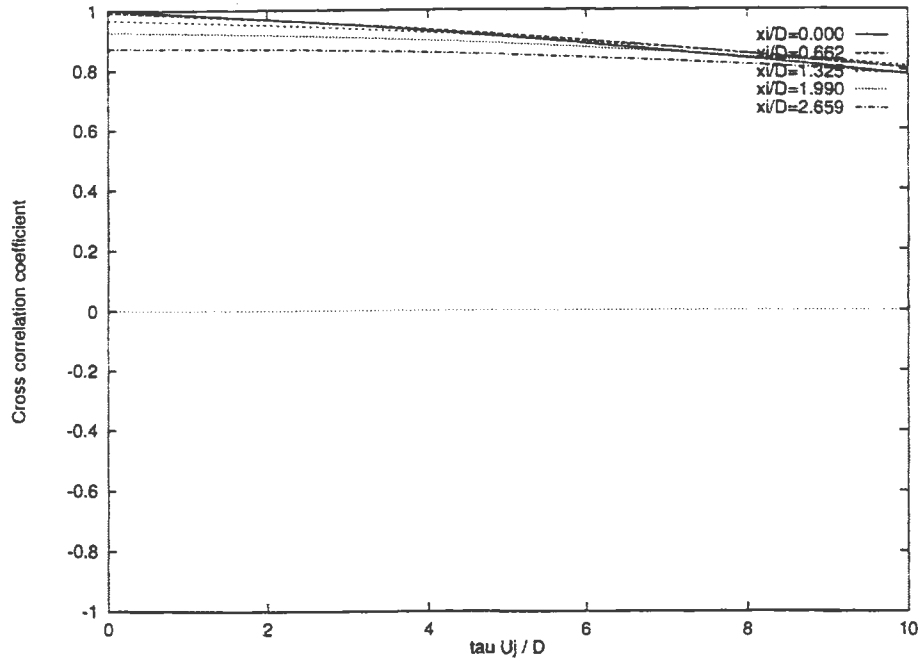


Figure 5.30: Cross Correlation for Run 4

Run 4 (Figure 5.30) once again shows significantly different results from the other runs. The correlation curves are stretched to the right even more so than in run 3. The global maximum of the  $\xi/D = 0.662$  curve occurs at  $\tau U_j/D = 0$ , resulting in an infinite convection speed. The other normalized convection speeds computed were much larger than expected, ranging from 2.513 at  $\xi/D = 1.324$  to 0.788 at  $\xi/D = 2.657$ . These results appear to be related to the transient phenomenon described in the discussion of run 4's radial velocity profiles in Section 5.3.1, where velocities near the inner wall of the nozzle accelerated to speeds higher than those observed at the center line. This is also the failure mode observed in runs 5 and 6.

Figure 5.31 shows a similar correlation used in Chu's investigation of a low speed jet [4]. While this is not directly comparable to the the correlations used for the

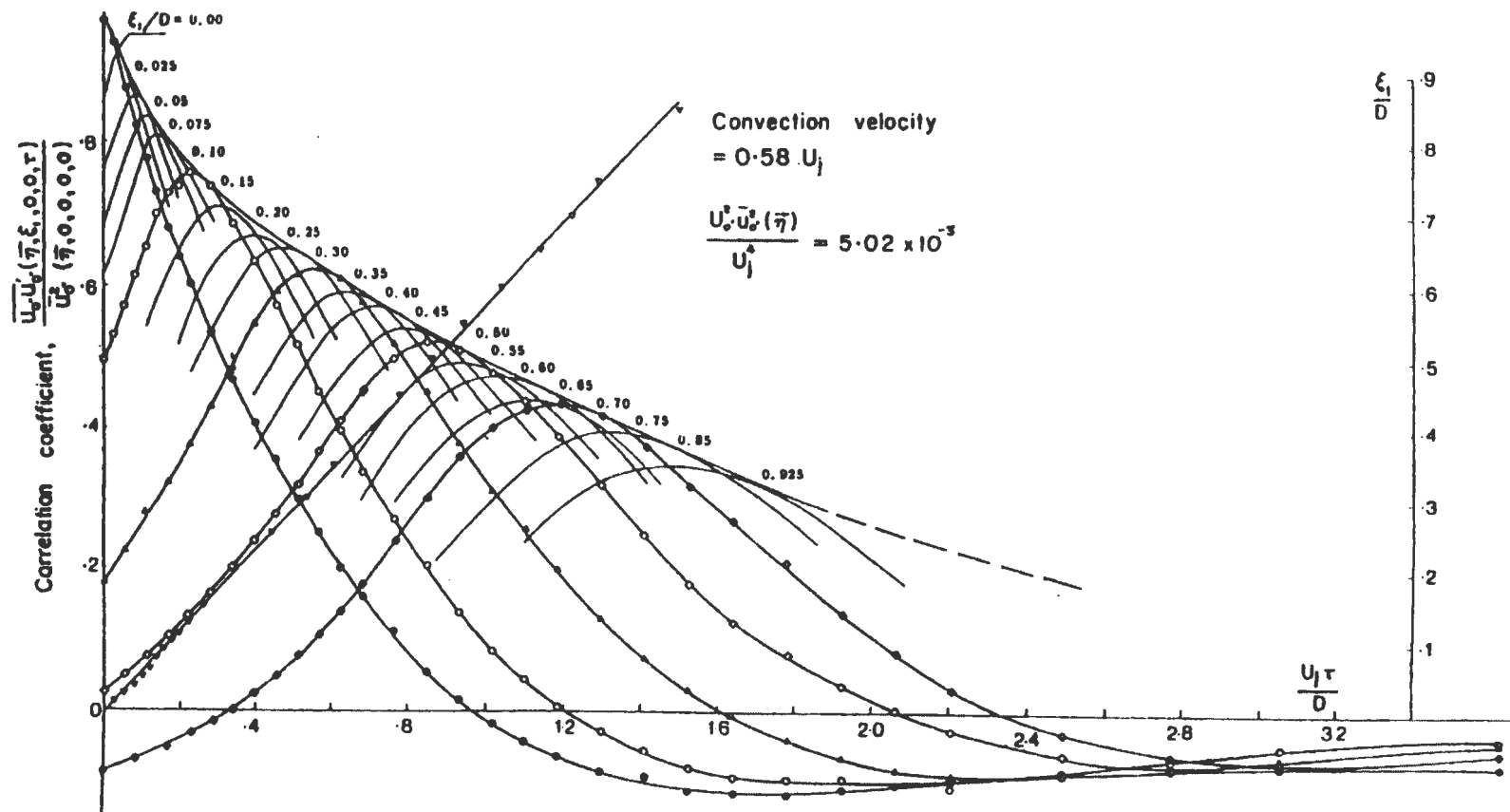


Figure 5.31: Cross Correlation for Low Speed Jet (from [4])

| $\xi/D =$ | 0.662     | 1.324     | 1.989     | 2.657     | Theory    |
|-----------|-----------|-----------|-----------|-----------|-----------|
| Run #     | $U_c/U_j$ | $U_c/U_j$ | $U_c/U_j$ | $U_c/U_j$ | $U_c/U_j$ |
| 1         | 0.393     | 0.395     | 0.401     | 0.398     | 0.468     |
| 2         | 0.516     | 0.511     | 0.520     | 0.526     | 0.479     |
| 3         | 0.520     | 0.514     | 0.509     | 0.508     | 0.492     |
| 4         | N/A       | 2.513     | 1.051     | 0.788     | 0.462     |

Table 5.5: Summary of  $U_c/U_j$  Predictions

calculation in this study due to the much lower jet speeds in Chu's study ( $\sim 400$  fps), it does confirm that the trends observed in correlation curves from the numerical results are quite similar to those observed experimentally. For instance, correlations in the Chu study show the same shift to the right as the separation distance is increased, and the shape of the individual correlation curves is similar to that observed in runs 1, 2 and 3.

## CHAPTER 6

### CONCLUSIONS AND FUTURE DIRECTIONS

#### 6.1 Conclusions

The unsteady flow from a three dimensional supersonic cylindrical jet has been computed numerically to investigate the physical processes which contribute to jet noise. The MacCormack explicit scheme is used to solve the 3D time-dependent Navier-Stokes equations. Scientific visualization techniques are used to investigate the spreading behavior of the jet and to describe the location and orientation of shock waves within the jet core. Time averages of the axial velocity and turbulence intensity are used to further describe the spreading behavior of the jet. A two point space-time cross correlation was used to estimate the convection speed in the shear layer.

From the results of these numerical simulations, several conclusions can be drawn. First and foremost, this study has shown that a three dimensional numerical simulation of jet mixing can capture several of the physical phenomena which contribute to jet noise. Second, this study has demonstrated the computer time and storage requirements needed to perform such a simulation. Third, this study has confirmed

that the formulation and implementation of boundary conditions can have a significant effect on the results of a numerical simulation.

The results of the simulations performed for this study capture some of the critical flow features which contribute to jet noise. One of these critical flow features is the shear layer; the growth of the shear layer determines how the jet spreads as it travels downstream of the nozzle exit. Another critical flow feature captured in the results of this study is the oblique shock system, which can influence both shock noise and screech.

The simulations performed for this study have also demonstrated the amount of resources required for a unsteady 3D simulation, both in terms of storage and computer time. For the  $141 \times 61 \times 38$  grid used in this study, 3.8 megabytes is required for the storage of the grid. A further 6.4 megabytes is required for the storage of each complete solution file. The running solver program requires approximately 32 megabytes of memory. While these requirements are relatively small by modern computing standards, the amount of compute time required for a time accurate solution is not. On a 100 megaflop processor, the solver required approximately 20 hours to complete 5,000 iterations. Considering that a time accurate simulation should be run for at least 75,000 time steps, this results in a minimum two week turnaround time between the start of the calculation and the start of post-processing on a 100 megaflop processor.

While 3D simulations of the type examined in this study have several advantages over 2D simulations of the same geometry, such as 3D pressure relief and the influence of 3D modes, these advantages must be weighed against the much higher computational resource requirements of the 3D simulation. A rectangular nozzle geometry

would likely benefit as much from a full 3D treatment as this study's cylindrical geometry.

The formulation and implementation of boundary conditions are among the most important and sensitive parts of the simulation process. In this study, the formulation and implementation of boundary conditions, particularly the outflow boundary condition, played a key role in the results of the simulation. The subsonic outflow condition used in this study can cause non-physical reflections back into the computational domain, while the supersonic outflow condition encourages the flow to remain supersonic beyond where it has been experimentally shown to have decelerated to subsonic speeds. Boundary conditions remain an area of active research in the simulation and analysis of jet noise.

While comparison with other results did not produce the desired degree of agreement, it is instructive in that it points to areas in which further research is required. For instance, the stretching of the axial turbulence intensity profile at the lip line relative to that observed in the laser velocimetry data from Yamamoto, et al. [32] indicates that the supersonic outflow condition used in these simulations is causing the flow to remain supersonic farther downstream than it should be. The rippling of the secondary compressions seen in the shock function visualizations shows that more work is needed in the formulation of a non-reflective center line condition. The lack of spreading observed in the entropy visualizations and velocity profiles indicate that further research into both grid refinement and outflow boundary conditions is needed.

## 6.2 Directions for Future Work

While this study has provided significant progress in the three dimensional simulation of cylindrical jets, there remain several avenues for future investigation. The following are issues for possible future research:

- Investigation of non-reflective boundary conditions for the center line and outflow boundaries, following the examples of al Qadi [1], Thompson [29], and Colonius, Lele, and Moin [6].
- Grid refinement, using either a 3D version of the fine grid of Kim, Kresja, and Khavaran [12] or a new grid specifically designed for an 3D simulation.

Of these, the issue of boundary conditions is the most pressing. The subsonic (fixed pressure) and supersonic (pressure extrapolation) outflow boundary conditions typically used in the calculations for this study do not adequately model the physics of a three dimensional unsteady jet with pressure waves propagating through the outflow boundary. To remedy this, non-reflective boundary condition formulations are required. The characteristic methods described by Colonius, Lele, and Moin as well as al Qadi [1, 6] show considerable promise, but their extension to a three dimensional coordinate frame is a nontrivial exercise. Thompson [29] has also formulated a three dimensional set of characteristic boundary conditions which also warrant further study.

Grid refinement is also an important avenue for future work. The grid used in this study was fairly coarse, particularly in the streamwise direction. This could be improved considerably, as Kim, Kresja, and Khavaran have shown in their later work [12]. Increasing the grid resolution will increase the natural dissipation of the

numerical scheme, which should improve the mixing rate and turbulence intensity predictions of the simulation. Additionally, the grid could be refined to model the trailing edge of the nozzle lip more closely. This region is treated as a blunt termination in the simulations done for this study, but in actuality the edge must have some finite radius. Al Qadi has shown that the lip geometry can have a significant impact on the mixing rate [1].

## BIBLIOGRAPHY

- [1] Ibraheem M. A. al Qadi. A numerical investigation the influence of jet lip geometry on mixing. Master's thesis, The Ohio State University, 1997.
- [2] Dale Anderson, John Tannehill, and Richard Pletcher. *Computational Fluid Mechanics and Heat Transfer*. Hemisphere Publishing, 1984.
- [3] A. Bayliss and L. Maestrello. Response of high subsonic jet to nonaxisymmetric disturbances. Technical report, NASA Langley Research Center, 1997.
- [4] Wing Chu. Turbulence measurements relevant to jet noise. Technical Report 119, UTIAS, November 1966.
- [5] S.G. Chuech, M.C. Lai, and G.M. Faeth. Structure of turbulent sonic underexpanded free jets. *AIAA Journal*, 28(11):549–559, November 1988.
- [6] Tim Colonius, Sanjiva Lele, and Parviz Moin. Boundary conditions for direct computation of aerodynamic sound generation. In *Proceedings of 14th Aeroacoustics Conference*, number 92-2075, Aachen, Germany, May 1992. DGLR/AIAA.
- [7] D.C. Fourguette, M.G. Mungal, and R.W. Dibble. Time evolution of the shear layer of a supersonic axisymmetric jet. *AIAA Journal*, 30(7):1123–1130, July 1991.
- [8] Richard Gallagher. *Computer Visualization*. CRC Press, 1995.
- [9] Richard G. Hindman. Generalized coordinate forms of governing fluid equations and associated geometrically induced errors. In *Proceedings of 5th Computational Fluid Dynamics Conference*, number 81-1008, Palo Alto, CA, June 1981. AIAA.
- [10] John D. Anderson Jr. *Introduction to Flight*. McGraw Hill, third edition, 1989.
- [11] Abbas Khavaran, Eugene Krejsa, and Chan Kim. Computation of supersonic jet mixing noise for an axisymmetric cd nozzle using k- $\epsilon$  turbulence model. NASA Technical Memorandum 105338, NASA Lewis Research Center, 1992.

- [12] Chan Kim, Eugene Krejsa, and Abbas Khavaran. Significance of shock structure on supersonic jet mixing noise of axisymmetric nozzles. *AIAA Journal*, 32(9), September 1994.
- [13] Arnold M. Kuethe and Chuen-Yen Chow. *Foundations of Aerodynamics*. John Wiley and Sons, fourth edition, 1986.
- [14] Pijush K. Kundu. *Fluid Mechanics*. Academic Press, 1990.
- [15] M.J. Lighthill. On sound generated aerodynamically. I: General theory. *Proceeding of the Royal Society of London*, A211:564–587, 1952.
- [16] M.J. Lighthill. On sound generated aerodynamically. II: Turbulence as a source of sound. *Proceedings of the Royal Society of London*, A222:1–32, 1954.
- [17] Reda Mankbadi, M. Ehtesham Hayer, and Louis Povinelli. Structure of supersonic jet flow and its radiated sound. *AIAA Journal*, 32(5):896–906, May 1994.
- [18] D. Papamoschou and A. Roshko. The compressible turbulent shear layer: an experimental study. *Journal of Fluid Mechanics*, 197:453–477, 1988.
- [19] Timothy Rozmajzl. Numerical investigation of screech for a 2D underexpanded supersonic rectangular jet. Master’s thesis, Ohio State University, 1997.
- [20] Hermann Schlichting. *Boundary Layer Theory*. McGraw-Hill, 1955.
- [21] William Schroeder, Ken Martin, and Bill Lorenson. *The Visualization Toolkit*. Prentice Hall, 1996.
- [22] James Scott. Numerical analysis of noise generation in unsteady jet flow. In *Proceedings of 27th Joint Propulsion Conference*, number 91-2254, Sacramento, California, June 1991. AIAA/SAE/ASME.
- [23] James Scott. Acoustic analysis using numerical solutions of the Navier-Stokes equations. In *Proceedings of 30th Aerospace Sciences Meeting and Exhibit*, number 92-0506, Reno, Nevada, January 1992. AIAA.
- [24] James Scott. Accuracy considerations in computational analysis of jet noise. In *Proceedings of 31st Aerospace Sciences Meeting and Exhibit*, number 93-0146, Reno, Nevada, January 1993. AIAA.
- [25] James Scott. A comparison of numerical and experimental results for unsteady flow associated with jet noise. In *Proceedings of 32nd Aerospace Sciences Meeting and Exhibit*, number 94-0459, Reno, Nevada, January 1994. AIAA.
- [26] James N. Scott. Private communication, December 1997.

- [27] Christopher Tam, K.K. Ahuja, and R.R. Jones III. Screech tones from free and ducted supersonic jets. *AIAA Journal*, 32(5), May 1994.
- [28] Christopher Tam and Dale Burton. Sound generated by instability waves of supersonic flows. Part 2: Axisymmetric jets. *Journal of Fluid Mechanics*, 138:273–295, 1984.
- [29] Kevin W. Thompson. Time-dependent boundary conditions for hyperbolic systems, II. *Journal of Computational Physics*, 89(2):439–461, 1990.
- [30] Pamela Walatka and Pieter Buning. *PLOT3D User's Manual*. NASA Ames Research Center, 1989.
- [31] Pamela Walatka, Jean Clucas, R. Kevin McCabe, Todd Plessel, and Rick Potter. *FAST User Guide*. NASA Ames Research Center, 1993.
- [32] K.J. Yamamoto et al. Experimental investigation of shock-cell noise reduction for single stream nozzles in simulated flight. Contractor Report 3845, NASA, 1984.