An Experimental and Computational Investigation of Turbulent Jet Impingement in a Confined Crossflow.

Keun Sun Chang
Louisiana State University and Agricultural & Mechanical College

Follow this and additional works at: https://digitalcommons.lsu.edu/gradschool_disstheses

Recommended Citation
https://digitalcommons.lsu.edu/gradschool_disstheses/4766

This Dissertation is brought to you for free and open access by the Graduate School at LSU Digital Commons. It has been accepted for inclusion in LSU Historical Dissertations and Theses by an authorized administrator of LSU Digital Commons. For more information, please contact gradetd@lsu.edu.
INFORMATION TO USERS

The most advanced technology has been used to photograph and reproduce this manuscript from the microfilm master. UMI films the text directly from the original or copy submitted. Thus, some thesis and dissertation copies are in typewriter face, while others may be from any type of computer printer.

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleedthrough, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send UMI a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

Oversize materials (e.g., maps, drawings, charts) are reproduced by sectioning the original, beginning at the upper left-hand corner and continuing from left to right in equal sections with small overlaps. Each original is also photographed in one exposure and is included in reduced form at the back of the book. These are also available as one exposure on a standard 35mm slide or as a 17" x 23" black and white photographic print for an additional charge.

Photographs included in the original manuscript have been reproduced xerographically in this copy. Higher quality 6" x 9" black and white photographic prints are available for any photographs or illustrations appearing in this copy for an additional charge. Contact UMI directly to order.

UMI
University Microfilms International
A Bell & Howell Information Company
300 North Zeeb Road, Ann Arbor, MI 48106 1346 USA
313 761 4700  800 521 0600
An experimental and computational investigation of turbulent
jet impingement in a confined cross flow

Chang, Keun Sun, Ph.D.
The Louisiana State University and Agricultural and Mechanical Col., 1989
An Experimental and Computational Investigation of Turbulent Jet Impingement in a Confined Cross Flow

A Dissertation

Submitted to the Graduate Faculty of the Louisiana State University and Agricultural and Mechanical College in partial fulfillment of the requirements for the degree of Doctor of Philosophy

in

the Department of Mechanical Engineering

by

Keun Sun Chang
B.S., M.E., Seoul National University, Seoul, Korea, 1979
M.S., M.E., Louisiana State University, Baton Rouge, LA, 1986
August 1989
Acknowledgement

The author wishes to express his sincere appreciation to his advisor, Dr. George D. Catalano, for his guidance, encouragement and patience over the last four and one-half years. It has been a great honor to the author to have completed a successful academic career and experimental development with his help.

The author wishes to thank his committee, Dr. T. W. Lester, Dr. V. A. Cundy, Dr. T. T. Charalampopoulos, and Dr. D. D. Reible, for their helpful suggestions and consistent concern through the author’s graduate studies.

The author expresses his gratitude to Ms. J. Mathis. During the course of this research, she was more than a research fellow. She was also a friend. The author thanks Mr. M. Kim not only for his friendship, but also for his help in the development of the computer expertise.

Finally, the author wishes to thank his parents, to whom he owes a great deal. This dissertation is dedicated to the author’s wonderful wife, Myree, and beloved son, Jess.

This research was supported by the United States Air Force, Wright Aeronautical Laboratories, Flight Dynamics Laboratory AFB, under Grant No. FY 1456-88-05065.
Contents

Acknowledgement ii
List of Tables v
List of Figures vi
Nomenclature vii
Abstract ix

1 Introduction 1

2 Background 3

3 Experimental System and Measurement Procedures 13
  3.1 Principles of Laser-Doppler Velocimeter 14
  3.2 Flow Configuration 17
  3.3 Optical Components of LDV System 18
  3.4 Signal Processing and Data Acquisition 23
  3.5 Data Analysis 28
    3.5.1 Mean, RMS, Skewness, and Flatness 29
    3.5.2 Reynolds Stress 32
    3.5.3 Correlations, Length Scales and Power Spectral Densities 32

4 Computational Approach 41
  4.1 Mathematical Model 42
  4.2 Solution Procedure 45
  4.3 Boundary Conditions 53
  4.4 Computational Details 60

5 Results and Discussions 63
  5.1 Boundary Conditions and Their Effects 64
  5.2 Mean Flow Fields 72
  5.3 Comparison of Calculated Results to Measurements 90
5.4 Turbulent Velocity Fields and Characterization ........................................... 93

0 Summary ................................................. 119

Bibliography ............................................. 122

Vita ......................................................... 127
List of Tables

3.1 The LDV parameters .......................................................... 23
4.1 The governing equations .................................................. 45
4.2 A summary of boundary conditions ................................. 60
5.1 Integral length scales and Taylor microscales for the free jet at y/D=1.0
and x/D=0.0 ........................................................................ 69
## List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>The problem to be considered</td>
<td>5</td>
</tr>
<tr>
<td>3.1</td>
<td>Principles of dual-beam LDV: (a) vector diagram, (b) fringe formation</td>
<td>15</td>
</tr>
<tr>
<td>3.2</td>
<td>The jet flow system</td>
<td>19</td>
</tr>
<tr>
<td>3.3</td>
<td>Laboratory schematic of the experimental system</td>
<td>20</td>
</tr>
<tr>
<td>3.4</td>
<td>Schematic of optical components of LDV system</td>
<td>22</td>
</tr>
<tr>
<td>3.5</td>
<td>Various types of Doppler signals: (a) detector signal ($I_D$-Doppler light intensity and $I_P$-pedestal light intensity), (b) band-pass filtered signal, (c) multi-particle signal</td>
<td>25</td>
</tr>
<tr>
<td>3.6</td>
<td>Various types of instantaneous velocity trace: (a) actual trace of turbulent velocity, (b) tracker output signal, (c) drop-out condition</td>
<td>27</td>
</tr>
<tr>
<td>3.7</td>
<td>Effects of sample size on statistical parameters (sampling rate = 2 KHz)</td>
<td>33</td>
</tr>
<tr>
<td>3.8</td>
<td>Comparison of autocorrelation functions between conventional average and FFT methods</td>
<td>38</td>
</tr>
<tr>
<td>3.9</td>
<td>Comparison of length scales between conventional average and FFT methods for various sample sizes</td>
<td>40</td>
</tr>
<tr>
<td>4.1</td>
<td>Grid arrangement: (a) two-dimensional view of staggered grid layout, (b) one-dimensional grid clustered for grid point $P$</td>
<td>48</td>
</tr>
<tr>
<td>4.2</td>
<td>Modification of the half-circular nozzle by rectangular cells</td>
<td>56</td>
</tr>
<tr>
<td>4.3</td>
<td>Mean velocity and mean temperature profiles at the jet exit plane for $R = 2$</td>
<td>57</td>
</tr>
<tr>
<td>4.4</td>
<td>Comparison of mean velocity, $U/U_o$, between two different jet exit boundary conditions in the plane of symmetry: (a) $R = 2$, (b) $R = 4$</td>
<td>59</td>
</tr>
<tr>
<td>5.1</td>
<td>Near field jet mean velocity and turbulence intensity profiles at $y/D=1.0$ and $x/D=0.0$</td>
<td>66</td>
</tr>
<tr>
<td>5.2</td>
<td>Autocorrelation functions of turbulent free jet in the potential core located at $y/D=1.0$</td>
<td>67</td>
</tr>
<tr>
<td>5.3</td>
<td>Auto-spectral densities of turbulent free jet in the potential core at $y/D=1.0$</td>
<td>68</td>
</tr>
</tbody>
</table>
5.4 Effects of the jet flow on the upstream boundary at x/D = -8.0 and z/D = 0.0; (a) mean velocity, (b) turbulence intensity ..................... 70
5.5 Integral length scale, L, and Taylor microscale, λT, for the upstream boundary at x/D = -0.8 and z/D = 0.0 ......................... 71
5.6 Maximum velocity trajectories in the plane of symmetry .................. 74
5.7 Similarity profiles of jet trajectories in the plane of symmetry : —

\( \frac{y}{D} = a\left(\frac{x}{D}\right)^b \) with a = 0.977, b = 0.911, and c = 0.335 ..... 75
5.8 Predicted scalar concentration, \( \frac{C}{C_j} \), and line of maximum maximum, \( S \), in x-y plane ; (a) R=2, (b) R=4, (c) R=6 ............. 77
5.9 Predicted scalar field, \( \frac{C}{C_j} \), in y-z plane ; (a) R=2, (b) R=4, (c) R=6 78
5.10 Total mean velocity vectors in x-y plane ; (a) R=2, (b) R=4, (c) R=6 81
5.11 Streamwise mean velocity isocontours, \( U/U_o \), in y-z plane ; (a) R=2,
(b) R=4, (c) R=6 ................................................................. 82
5.12 Isometric view of the interaction of jet and crossflow in x-direction for R=2 ................................................................. 84
5.13 Isometric view of the interaction of jet and crossflow in y-direction for R=2 ........................................................................ 85
5.14 Isometric view of the interaction of jet and crossflow in x direction for R=4 ........................................................................ 86
5.15 Isometric view of the interaction of jet and crossflow in y-direction for R=4 ................................................................. 87
5.16 Streamwise mean vorticity contours at two downstream locations in y-z plane ; (a) R=2, (b) R=4, (c) R=6 ................................. 89
5.17 Comparison of mean velocities between predictions and measurements in x-y plane for R = 2 ; •, measurement, —, prediction .......... 91
5.18 Comparison of mean velocities between predictions and measurements in x-y plane for R = 4 ; •, measurement, —, prediction .... 92
5.19 Normalized Reynolds stress distributions in x-y plane for R=2 ; symbols, measurement, — in (c), prediction ......................... 95
5.20 Normalized Reynolds stress distributions in x-y plane for R=4 ; symbols, measurement, — in (c), prediction ......................... 96
5.21 Comparison of dissipation rate between predictions and measurements for R = 2 ; (a) x/D=4.0, (b) x/D=8.0 ; •, measurement, —, prediction .................................................. 98
5.22 Comparison of dissipation rate between predictions and measurements for R = 4 ; (a) x/D = 4.0, (b) x/D = 8.0 ; •, measurement, —, prediction .................................................. 99
5.23 Autocorrelation functions for R=2 at z/D = 0.0 ............................. 102
5.24 Autocorrelation functions for R=4 at z/D = 0.0 ............................. 103
5.25 Auto-spectral densities for R=2 at z/D = 0.0 ............................. 105
5.26 Auto-spectral densities for R=4 at z/D = 0.0 ............................. 106
5.27 Integral time scales at z/D = 0.0 ; (a) R=2, (b) R=4 ............................. 108
5.28 Integral length scales at $z/D = 0.0$; (a) $R=2$, (b) $R=4$ .................................. 109
5.29 Taylor microscales at $z/D = 0.0$; (a) $R=2$, (b) $R=4$ .................................. 110
5.30 Measured probability distributions of $x$-component velocity at $x/D=4$
and $z/D=0.0$ for $R=2$; (a) $y/D=2$, (b) $y/D=4$, (c) $y/D=6$ ............. 114
5.31 Measured probability distributions of $x$-component velocity at $x/D=4$
and $z/D=0.0$ for $R=4$; (a) $y/D=3$, (b) $y/D=5$, (c) $y/D=7$ ............. 115
5.32 Measured skewness factor profiles of $x$-component velocity at $x/D=4.0$
and $z/D=0.0$ for $R=2$ and $4$ ......................................................... 116
5.33 Measured flatness factor profiles of $x$-component velocity at $x/D=4.0$
and $z/D=0.0$ for $R=2$ and $4$ ......................................................... 117
Nomenclature

\( C \)  
mean species concentration

\( C_j \)  
mean species concentration at the jet entrance

\( C_\mu \)  
constant in turbulence model

\( D \)  
jet diameter at the injection

\( E(k) \)  
3-D energy spectrum function

\( F_u \)  
flatness factor

\( H \)  
channel spacing

\( I \)  
integer, \( I = N/M \)

\( L \)  
longitudinal integral length scale

\( M \)  
number of time delay in autocorrelation function

\( N \)  
number of velocity samples

\( P \)  
effective pressure, \( p + \frac{2}{3} \rho k \)

\( R \)  
velocity ratio \( (V_j/U_0) \)

\( Re \)  
Reynolds number \( (\frac{V_j D}{\nu}) \)

\( R(\tau) \)  
autocorrelation function

\( S_\phi, S_C, S_P \)  
source terms in the governing equations \( (S_\phi = S_C + S_P \Phi_P) \)

\( S(f) \)  
auto-spectral density function

\( S_u \)  
skewness factor

\( T \)  
integral time scale

\( U, V, W \)  
mean velocity components

\( U_0 \)  
cross stream velocity (tunnel flow speed)

\( V_j \)  
jet injection velocity

\( c_1, c_2 \)  
constants in turbulence model

\( d_f \)  
fringe spacing

\( d_p \)  
probe volume diameter

\( f \)  
frequency in auto-spectral density function

\( f_d \)  
Doppler frequency

\( f_1, f_2 \)  
scattered frequencies of laser beams

\( f_{01}, f_{02} \)  
frequencies of laser beams

\( f_s \)  
frequency shift of laser beam

\( f(\tau) \)  
spatial correlation function

\( k \)  
turbulent kinetic energy
$k$  wave vector  
$l_{pv}$  probe volume length  
$p$  pressure  
$u, v, w$  fluctuating velocity components  
$u(t)$  instantaneous velocity  
$\hat{u}, \hat{v}, \hat{u}\hat{v}$  turbulent shear stress  
$x, y, z$  Cartesian coordinates  
$y^+$  dimensionless value of $y$ ($= \rho k^{1/2} c_\mu^{1/4} y / \mu$)  

$\Gamma_\Phi$  diffusion coefficient of $\Phi$  
$\Phi$  general dependent variable  
$\Phi_P$  general dependent variable at grid point P  
$\Omega_x$  $x$-direction mean vorticity  

$\epsilon$  dissipation rate of $k$  
$\theta$  beam intersection angle  
$\lambda$  wavelength of laser beam  
$\lambda_T$  Taylor microscale  
$\lambda_*$  underrelaxation factor  
$\mu$  molecular viscosity  
$\nu$  kinematic viscosity  
$\rho$  density  
$\sigma_T, \sigma_k, \sigma_e$  turbulent Prandtl/Schmidt numbers  
$\tau$  delay time in autocorrelation function
Abstract

An experimental and computational investigation is made for the turbulent flow field generated by a jet discharging transversely into a confined moving stream. For jet to cross flow mean velocity ratios (R) equal to 2 and 4, measurements are made using a laser-Doppler velocimeter (LDV) in conjunction with a tracker processor. Quantities measured include mean velocities, turbulence intensities, turbulent shear stress, and dissipation rates of the turbulent kinetic energy. The jet flow Reynolds numbers investigated are $1.5 \times 10^4$ for $R=2$ and $3.0 \times 10^4$ for $R=4$.

The structured nature of the turbulent flow field is documented from the statistical measurements. The autocorrelation functions are calculated from the instantaneous velocities using a direct Fourier transform method. The auto-spectral density functions are calculated via a Fast Fourier Transform of the autocorrelation functions. The integral length scale and the Taylor microscale are calculated from the autocorrelation functions using the Taylor hypothesis. The probability density function, skewness and flatness factors are calculated from the randomly sampled instantaneous velocities.

For the velocity ratios (R) equal to 2, 4 and 6, calculation results are obtained by
solving the steady, three-dimensional elliptic forms of the Reynolds time-averaged equations. The finite-difference solution scheme is employed for the calculation procedure. The Reynolds stresses appearing in the time-averaged equations are calculated from the two equation k-ε model of turbulence.
Chapter 1

Introduction

Turbulence is often referred to as the last unsolved problem in Newtonian mechanics. Turbulent motion, a flow condition in which the dependent field variables such as velocity, pressure and temperature are random both in space and time, is receiving an ever growing degree of interest among engineers and scientists. In fact, the overwhelming percentage of flows which occur in Nature or are created by Man is turbulent. Since a deterministic approach to turbulent motion is impossible due to its randomness, research in turbulent fluid mechanics has resorted to the use of experimental techniques and computational schemes.

Recent advances in diagnostic techniques have injected new momentum and excitement into turbulence research. As a consequence, a number of new aspects and insights into turbulent flows have been discovered. The existence of coherent structures is one good example. Such a discovery in turbulent flows has led to a fundamental reexamination of our understanding of turbulence. It is now commonly accepted that large-scale coherent structures play important roles in energy
production, mixing and noise generation. The existence of coherent structures has been extensively reviewed by many investigators\cite{1-4}.

A number of turbulence modeling techniques together with improved numerical methods have been used as tools for the analysis of many engineering problems. Numerical methods allow systematic variations of boundary conditions and geometric variables, and also provide information on quantities of interest simultaneously and economically in cost and time. However, numerical methods cannot yet resolve details of flow physics such as scales of turbulence and other time-evolving quantities at practically occurring Reynolds numbers\cite{5}. Numerical methods in turbulence research supplement, rather than replace, experimental measurements.

In this study, a combined experimental and computational approach is made for the investigation of the turbulent flow field that results from the introduction of a jet transversely into a confined moving stream. The flow field is a basic configuration encountered in many real engineering problems such as V/STOL aerodynamics, the design of gas turbine combustors, the internal cooling of turbine blades, and hazardous waste disposal into bodies of water or the atmosphere. The ultimate objective of this study is to increase the understanding of the jet in a confined cross flow.
Chapter 2

Background

The flow of a turbulent jet issuing into a cross stream is a very complex one. The flow field is generally fully three-dimensional. The literature review presented in this chapter reveals some of the essential features of the jet in a cross flow, including the following: (a) the transition of the jet trajectory from an initially vertical jet to one which becomes parallel with the cross flow; (b) the production of a vortex pair in the kidney shaped cross-section; and (c) the creation of a bluff-body type wake in the cross flow (Figure 2.1). If the jet impinges on the opposite wall in the confined configuration, additional complexities arise in the flow field.

Any attempt to attain a clearer understanding of these flows poses a great challenge for both the experimental techniques and prediction methods. Experience and developed insight and ingenuity are often required on the part of the researcher. Consider first the experimental challenge. Conventional measurement techniques such as pressure or hot-wire probes may not provide sufficiently accurate information. With conventional measurement techniques, the flow field with severe
pressure gradients and strong stream curvatures is difficult to document. These types of flows are extremely sensitive to local geometry and probe interference. Problems associated with turbulent structure measurements are even more acute because the conventional measurement techniques cannot follow the high turbulence fluctuations of the flow field accurately. Accordingly, a laser-Doppler velocimeter (LDV) technique is often utilized.

The LDV is growing in use for fluid measurements. It has distinct and substantial advantages over the conventional measurement techniques. One disadvantage is its expensive cost. The most outstanding merit of the LDV is the fact that it is a non-intrusive measurement. The LDV technique with its inherent linear sensitivity allows accurate documentation of highly turbulent flows and recirculating flows without disturbing the flow field. Another important merit of the LDV application is in the faithful response of the LDV probe to the instantaneous velocities. This factor is essential for the measurements of length scales or power spectral densities. An extensive review of the LDV principles and applications to many types of flow is provided by Durst et al.[6].

Consider next the prediction challenge. Three possible prediction approaches are reviewed. The first of these is the application of integral methods. These methods require many simplifying assumptions dealing with entrainment functions and similarity profiles of the jet cross-section. The integral methods are only capable of modeling global effects, and thus are not considered here. The second approach
Figure 2.1: The problem to be considered
is the application of the inviscid flow model (vortex model). The model is effective simulating the propagating motion of the vortex in a Lagrangian frame. The details of turbulent structures, however, cannot be resolved with this inviscid model.

The third prediction method, which is utilized in the computational portion of this study, is the application of a numerical solution method together with a comprehensive turbulence modeling technique. While a number of turbulence models have been developed previously and may be applicable to this problem(7,8), the two-equation $k-\epsilon$ model of turbulence is employed. The objectives of this computational study are twofold: (1) to determine the extent to which the present flow field can be represented by a computational method based on time-averaged equations and turbulence modeling techniques; and (2) to complement the experimental data.

Most of the previous experimental work has centered on a single jet in an unbounded cross flow. Keffer and Baines[9], Kamotani and Greber[10], and Ramsey and Goldstein[11] provided measurements of jet trajectories, velocities and temperature profiles. It was observed that the jet velocity and temperature trajectories for a heated jet are mainly determined by the velocity ratio (or momentum ratio) and weakly dependent upon the density ratio. McMahon et al.[12], Chassaing et al.[13], and Moussa et al.[14] have measured the near field of the jet and observed that the noteworthy feature of the velocity field is the presence of a vortex pair in the kidney shaped cross-section. In view of recent advances in velocimetry, the velocity
measurements reported, obtained by pressure or hot-wire probes, must be considered of uncertain precision due to the inability of this instrumentation to accurately document the reverse and highly turbulent flows in the near field regions.

The only measurements using the laser-Doppler velocimeter in this flow problem were reported by Crabb et al. [15], who also provided an extensive review of previous work in this flow field up to 1980. The LDV was used in the near field (x/D < 6) to measure the mean velocity, turbulence intensities and shear stress for the velocity ratios of R = 1.15 and 2.3. Turbulence intensities reported in their measurements are extremely high, especially in the wake region (a maximum of local turbulence intensities is 75% at x/D = 0 for R = 2.3). Crabb et al. observed that locus of maximum velocity does not necessarily correspond to fluid from the jet exit, that is, the downstream regions of higher velocity are composed of mainly free stream fluid. Further investigations are necessary to verify this result.

For low velocity ratios (R < 2), the turbulent kinetic energy and the three components of the shear stress were measured by Andreopoulos and Rodi [16] using a three-sensor hot-wire probe. Their measurements, supported by the visualization study of Foss [17], corroborate the essential features of a jet in crossflow, previously reported by other investigators [9, 10, 11, 15]. The velocity fluctuation statistics and the velocity-temperature fluctuation statistics were reported by Andreopoulos [18, 19]. One important observation from the results of this investigator is the fact that, at low velocity ratios (R < 2), the mean velocity and the mean temperature profiles,
which were initially fully-developed pipe flows, are found to be considerably skewed downstream at the jet exit plane, indicating interactions between the jet stream and the cross stream prior to the jet exhaust.

Research of relevance to V/STOL aerodynamics has been performed mainly by NASA Langley research groups[20-22] with the review of Skifstud[23] summarizing the progress until 1970. The primary interest is in the pressure distribution on the plate through which the jet exhausts. It is well known that, when a V/STOL aircraft is in transition phase, the interaction between the jet stream and the main stream induces a pressure loading on the surface, which results in significant effects on lift and pitching moment of the aircraft. Recent works by Fearn and Weston[24] and Taylor and Watkins[25] extended earlier investigations in this field by including the variational effects of the jet angle and velocity ratios.

Few measurements have been reported for the confined configuration. Stoy and Ben-Haim[26] reported measurements of jet trajectories for a single blockage ratio (H/D=3.05) and provided empirical correlations of the impingement point in terms of the velocity ratios for 2.5 ≤ R ≤ 7.0. Their measurements confined attention to ranges of parameters applicable to the impingement cooling of turbine blades. Holdeman and Walker[27] investigated a geometry relevant to diffusion air jets in gas-turbine combustion chambers. Their measurements encompassed temperature fields for both a single jet and a row of jets in a confined crossflow. These data were then used to develop correlations characterizing the behavior of the jet in terms of
flow and geometric variables. Kamotani and Greber[28] presented both velocity and temperature measurements of a single jet, a row of jets and two opposing jets in a crossflow. It is the author's opinion that no documentation of turbulent quantities for the confined problem with impingement has been reported in the literature.

The theoretical work addressing this problem has mainly involved application of integral methods[29,30]. The methods are based on the similarity assumptions of the velocity profile across the jet cross-section so that the original partial differential equations can be reduced to ordinary ones. The integral methods are only applicable to situations where the discharge creates the jet-type flow, that is, the jet's cross-sectional shape is not significantly altered by walls or free surfaces. The global effects of turbulence can be simulated in these methods, using further semi-empirical inputs regarding the entrainment law and the jet cross-sectional shape (usually assumed to be circular). The latter restriction has been removed by Adler and Baron[31] who succeeded in predicting a kidney-shaped jet cross-section with their integral approach. The methods are well justified only if the gross features of jet development such as jet trajectories or the jet’s penetration distance into the surrounding medium are considered (not the viscous effect or entrainment).

Another predictive method applied to this problem is the application of potential flow theory[32–34]. In this approach, the jet is replaced with two counter-rotating vortices or with a sink-doublet singularity distribution. The motivation for their
inclusion in the shear flow is to simulate the rolling-up motion of vortices using a Lagrangian point of view [35]. The method also yields several important variables with sufficient accuracy, including the surface pressure distribution and vortex spacing and trajectories. The most significant drawback to this model is the neglect of viscous effects. Karagozian [36] included the viscosity by replacing the vortex structures with two Rankine vortices (solid-body rotation plus potential vortex), but this crude inclusion of the viscosity cannot properly account for the dominant phenomena of viscous action in this flow field.

The predictive methods described adapt highly idealized flow models and require empirical inputs with too many assumptions and approximations for the methods to possess much breadth of applicability. Along with the development in numerical methods, it is now possible to use a sophisticated turbulence model to obtain numerical results for this complex flow. The important requirements for a useful model are such that it be relatively simple yet capable of accounting for most phenomena with minimum approximations and assumptions. While a number of turbulence models have been developed and tested previously [7], the two-equation k-ε model of turbulence is the most useful for this problem.

Calculation methods of the numerical type (finite-element or finite-difference) applied to jets in a crossflow are reported by several investigators. Patankar et al. [37] employed a finite-difference scheme together with the two-equation turbulence model to compute the single jet in a crossflow with relatively large values
of R (2 to 10). Their computation of mainflow characteristics like jet trajectories, compared to experiments of Keffer and Bains[9] and Ramsey and Goldstein[11], was fairly good, but sufficient resolution was not possible for the axial velocity profiles. Bergles et al.[38] and Tetchell[39] applied a calculation method similar to that of Patankar et al.[37] to calculate the flow from a row of jets in a crossflow. Since Bergles et al. and Tetchell were interested primarily in the external cooling problem, they considered very low velocity ratios (R ≤ 0.3). Thus, numerical schemes in these two literatures were restricted to cases where no negative velocities occurred. Tatchell[39] showed that the parabolic method failed to give realistic results even for the velocity ratio R as low as 0.1. Rodi and Srivasta[40] developed a partially-parabolic finite-difference calculation procedure in which the regions of reverse flow are handled using a locally elliptic procedure. The method is more economical in computer time and particularly in computer storage than a fully three-dimensional elliptic method, but the applicability of the numerical scheme is again restricted to low velocity ratios (R ≤ 0.3).

Fully three-dimensional elliptic finite-difference schemes are reported by Khan et al.[41] and Demuren[42] for the calculation of a row of jets in the confined situation. Khan et al. discussed the fact that the k-ε model did not adequately resolve the high complexity of turbulence in the wake region and the results were grid independent in the initial region (x/D ≤ 4.0). Recently, Sykes et al.[43] applied a quasi-equilibrium version of the Reynolds-stress closure model to the unconfined
problem. The model uses a very crude estimation of the length scale as \( L = 0.25 \, D + 0.025 \, x \). This relation indicates that the length scale is only dependent on the axial distance from the jet discharge hole and independent of the vertical distance. The length scale measurements of this study (presented in Chapter 5) show significant variations along the vertical distance. Consequently, the prediction of velocity and scalar field in their results[43], compared to experimental data of Andrepoulos and Rodi[16], is not precise. This numerical method, however, effectively illustrates the dependence of vorticity dynamics on velocity ratios. Sykes et al. demonstrated that, for a large velocity ratio, the source of the streamwise vorticity can be readily traced back to the original streamwise vorticity in the sides of the vertical jet.

The review of the previous experimental and numerical work clearly reflects both the practical importance of the flow situation and the difficulty of providing a comprehensive treatment either analytically or experimentally. This work extends the previous work. Particular emphasis will be placed on the effects of the confining surface on the flow field, which has been heretofore ignored.
Chapter 3

Experimental System and Measurement Procedures

Traditionally, researchers in experimental fluid mechanics have relied on intrusive probes to obtain information on quantities of interest, such as velocity and pressure. Pressure probes or hot-wire probes are still undoubtedly important tools for analyzing many fluid dynamic problems. However, as the challenges facing the experimenter grow with increasing turbulent fluctuations, and even more acutely as the levels of fluid dynamic interest grow and become sophisticated, the conventional measurement techniques with the heuristic and limited applicability are now being replaced by a new measurement technique with distinct and substantial advantages. This experimental study is thus concerned with a non-intrusive, optical method (Laser-Doppler Velocimetry, LDV) for the measurement of instantaneous velocities.
3.1 Principles of Laser–Doppler Velocimeter

The development of the continuous gas laser made possible the use of the Doppler effect as an optical, non-intrusive method for measuring the velocity for gases, liquids and solids. The optical technique represented by the formation of interference patterns of laser light allows the determination of particle velocity by measuring the transit time of the particle across a known number of interference fringes. This technique, which can be used in a number of different ways, is referred to as laser–Doppler velocimeter (LDV). The first application in fluid mechanics was described by Yeh and Cummins[44] in 1964, who measured velocities in a fully–developed laminar pipe flow of water.

The LDV optical arrangement used in this study is the dual–beam mode. Two intersecting light beams of equal intensity produce a fringe pattern within their volume of intersection (Figure 3.1). The distance, \( d_f \), between fringes is given by the relationship:

\[
d_f = \frac{\lambda}{2 \sin(\theta/2)}
\]  

(3.1)

where \( \lambda \) is the wavelength of the light and \( \theta \) is the angle between the two beams.

The laser beams may be described as plane waves with a Gaussian distribution of light intensity at the beam cross-section. Suppose that two laser beams with frequencies of \( f_{01} \) and \( f_{02} \) propagate in two different directions, \( \hat{e}_1 \) and \( \hat{e}_2 \), respectively. A small particle passing through the intersection of the beams will scatter light, modulated with a frequency value dependent on the scattering direction and parti-
Figure 3.1: Principles of dual-beam LDV: (a) vector diagram, (b) fringe formation

\( d_f = \text{fringe spacing} \)

\( d_p = \text{probe volume diameter} \)

\( l_p = \text{probe volume length} \)
cle velocity. The two shifted frequencies from the two laser beams in the scattering direction \( \hat{e}_s \) can be expressed in terms of the particle velocity \( \vec{V}_p \), the scattering direction \( \hat{e}_s \), and the wave length of the laser beams \( \lambda \):

\[
\begin{align*}
 f_1 &= f_{01} + \frac{\vec{V}_p \cdot (\hat{e}_s - \hat{e}_1)}{\lambda} \\
 f_2 &= f_{02} + \frac{\vec{V}_p \cdot (\hat{e}_s - \hat{e}_2)}{\lambda}
\end{align*}
\tag{3.2}
\tag{3.3}
\]

The detector actually measures the difference between the two shifted frequencies, \( f_1 \) and \( f_2 \). The difference frequency, \( f_d(= f_1 - f_2) \), is called the Doppler frequency and is given by :

\[
f_d = f_s + \frac{\vec{V}_p \cdot (\hat{e}_2 - \hat{e}_1)}{\lambda}
\tag{3.4}
\]

where \( f_s(= f_{01} - f_{02}) \) is the frequency shift between the two incident beams. The frequency shift \( f_s \) can be set at a desired value by a Bragg cell unit placed in the optical system. The frequency shift enables a determination of the flow direction.

In terms of the velocity normal to the laser axis and the intersection angle of two beams, the Doppler frequency may be rewritten :

\[
f_d = f_s + \frac{2V_z \sin(\theta/2)}{\lambda}
\tag{3.5}
\]

The above relationship implies that the LDV system has a unique response to the fluid velocity. The measurement is based on the stability and linearity of optical electromagnetic waves and is only weakly affected by other physical parameters such as temperature and pressure. However, as in the case of its main competitor, the hot-wire anemometer, the application of the LDV to turbulence measurements is
not trivial. This is due to sources of LDV errors in turbulence measurements. The difficulties facing experimenters grow with increasing turbulent fluctuations. Typical examples of errors are phase fluctuations of particles and velocity biasing errors due to nonuniformly distributed particle concentrations. The principal sources of LDV errors have been reviewed by Buchhave et al. [45].

3.2 Flow Configuration

The experiments are performed in a subsonic wind tunnel in the Experimental Fluid Dynamics Laboratory of the Mechanical Engineering Department at Louisiana State University. The dimensions of the test section are 60 cm wide, 45 cm high, and 180 cm long. The nearly uniform flow in the test section is attained by routing the airflow through a 12:1 contraction section and flow straightening honeycomb tubes. The free stream turbulence intensity is less than 0.8% in the range of tunnel velocities (9.5–50.0 m/s). Optical access for the LDA measurements is provided through a removable plexiglass wall in the test section.

The jet stream is supplied from the laboratory's compressed air line and adjusted by a high-precision pressure regulator. In order to minimize the effect of the tunnel wall boundary layer, the jet exit is mounted flush in a flat plate located 12 cm above the bottom wall of the tunnel and the jet is aligned with the test section center line. A top plate is similarly mounted inside the test section of the tunnel, and its distance from the wall is adjusted such that \( \frac{H}{D} = 10 \). The plates, which are made of 60 cm wide, 180 cm long, and 0.6 cm thick plexiglass, have been carefully
contoured at the leading edges to insure smooth transition of the tunnel flow.

Special care is taken to obtain a flat velocity profile and low turbulence intensity at the nozzle exit. This is accomplished by use of a settling chamber, a fine mesh screen and a smooth shaped contraction section with the contraction ratio of 16:1 through a 6.7 cm length (Figure 3.2). Variation of the mean velocity ratio values \( R \) is achieved by adjustment of the compressed air line pressure regulator while keeping the tunnel flow speed at 9.5 m/s. The maximum obtainable jet velocity is approximately 50.0 m/s and the corresponding Reynolds number based on the 1.27 cm jet diameter is \( 3.8 \times 10^4 \). The Reynolds numbers investigated are \( 1.5 \times 10^4 \) for \( R=2 \) and \( 3.0 \times 10^4 \) for \( R=4 \). The assumption of the symmetry of the entire flow field about plane, \( z=0 \), is validated. The jet flow and the tunnel flow are seeded independently with olive oil particles by use of an aerosol generator for tunnel flow and a pneumatic tool lubricator for jet flow. A laboratory schematic of the entire experimental system is presented in Figure 3.3.

### 3.3 Optical Components of LDV System

The LDA optical arrangement has been constructed from Dantec 55X Modular Optics components (Figure 3.4). A Spectra Physics Model 106-1 He-Ne laser rated at 15 mW provides a monochromatic, coherent light source of wavelength \( \lambda = 632.8 \) nm and beam diameter 0.68 mm at 1/e² points. The beam splits into two parallel beams of equal intensity by passing through a beam splitter module. The frequency of one beam is upshifted 40 MHz by the Bragg cell which is driven by the frequency
Figure 3.2: The jet flow system
Figure 3.3: Laboratory schematic of the experimental system
shifter. The other beam is displaced by the displacer module. The shifted frequency causes the interference fringes in the probe volume to move with a velocity \( V_f = 40 MHz \times d_f \), where \( d_f \) is the fringe spacing. This capability allows the user to determine the direction of the flow.

The emerging unshifted and shifted beams pass through the backscatter section and are fed to the beam translator, which adjusts the intersection angle by reducing the beam separation distance from 60 mm to 13 mm - 39 mm. The beams then proceed through the beam expander which expands the parallel incoming beams by a factor, 1.95 and thereby decreases the size of the probe volume by the same factor, approximately quadrupling the light intensity. The beams leave the beam expander and pass through a 600 mm focal length lens which focusses them to a point. When a beam separation distance of 39 mm is selected at the beam translator, the beam separation distance of 76 mm at the front lens and a intersection angle of 7.25° result.

The light scattered from seeding particles passing through the probe volume is collected in the backward direction by the photomultiplier (PM) optics module. This module, with a 150 mm focal length lens, focuses the collected light into a pinhole aperture of 0.1 mm diameter. The PM section which acts as a spatial filter converts the photon flux to an electric signal.

The optical set-up described is a one-component backward scatter system operating in the fringe-Doppler mode. The assembled LDV system can rotate 360
Figure 3.4: Schematic of optical components of LDV system
degrees as well as traverse in three-dimensions.

The LDV actually measures the instantaneous velocity of small particles suspended in the flow. Hence, the particles must be small to follow the local velocity of the flow. As mentioned before, the wind tunnel flow and the jet stream are seeded with olive oil particles. For the seeding method used, the mean oil droplet diameter is estimated to range from 0.8–3.0 \( \mu \)m. This particle diameter range is appropriate to follow air flows where turbulence frequencies exceed 1 KHz[6]. The LDV parameters for velocity measurements are summarized in Table 3.1.

### 3.4 Signal Processing and Data Acquisition

A method for signal processing and data acquisition is vital in LDV application. The output signal from the PM consists of the sum of the Doppler frequency representing the velocity and the 40 MHz shifted frequency. In order to achieve optimum resolution and noise filtering in the signal analysis equipment, the signal is fed to the mixer (frequency shifter) to be electronically down shifted. The effective shift
of the Doppler frequency $f_*$ can be selected to a desired level from $\pm 10$ KHz to $\pm 9$ MHz.

Characteristics of the Doppler signal are shown Figure 3.5. In Figure 3.5 (a), the envelope (top dotted line) indicates the Gaussian light intensity distribution in the probe volume and the solid line is the DC component of light intensity (pedestal). The drop and rise pattern of light intensity in Figure 3.5 (a), resulting from the consecutive bright and dark fringes, contains the frequency information from which the velocity information is extracted.

The electronic mixer output signal is analyzed by a TSI Tracker Type 1090. The signals are first amplified to an optimum signal-to-noise ratio and passed through selectable band pass filters prior to the signal analysis. The high pass filters remove the DC component of the signal (pedestal) and the low pass filters remove the high frequency noise. The band-pass filtered Doppler signal is shown in Figure 3.5 (b). The tracker is a phase locked loop (PLL) device. The processor locks on to the Doppler frequency and continues to track the instantaneous Doppler frequency as long as the internal servo-loop stays locked. Detailed interpretations of the PLL principle may be found in the references[6,46].

The continuous nature of the tracker output allows the mean and rms quantities to be read directly on appropriate meters. Figure 3.6 (b), from reference[47], shows the continuous velocity trace from the tracker output. The signal usually contains high frequency noise. The noise is primarily due to the phase fluctuations of particles
Figure 3.5: Various types of Doppler signals: (a) detector signal ($I_D$—Doppler light intensity and $I_P$—pedestal light intensity), (b) band-pass filtered signal, (c) multi-particle signal.
in the probe volume (called the ambiguity noise). The errors in the mean and rms velocities due to the noise are not significant [45]. However, at a certain noise level, the tracker cannot lock on to the instantaneous Doppler frequency. The term "drop-out" refers to a condition in which the tracker is not locked on to the instantaneous frequency of the Doppler signal. The drop-out also arises in sparsely seeded flows (in high speed air flows). Fortunately, when the tracker enters the drop-out condition, a protection circuit of the tracker (sample and hold circuit) is used to lock the loop on the last measured frequency and hold the frequency until a valid Doppler signal is retrieved (see Figure 3.6 (c)). Although continuous data output is recovered in this way, the errors in the mean and RMS of the output are present as well as the missing of the high frequency end of the drop-out signal. These errors are referred to as statistically biased errors. Dependent on the flow condition and measurement location, the velocity signal in this study exhibits either the continuous condition with high frequency noise shown in Figure 3.6 (b) or the drop-out condition in Figure 3.6 (c). The drop-out signal generally arises in highly sheared regions where the rapid transition between the jet stream and the cross stream occurs.

The errors can be significantly reduced by appropriate correction methods using computer analysis. Since computers operate on discrete data points, the resulting data can be weighted by the time duration which is held between successive Doppler signals (will be discussed in the next section). The probability density function,
Figure 3.6: Various types of instantaneous velocity trace; (a) actual trace of turbulent velocity, (b) tracker output signal, (c) drop-out condition.
autocorrelation and spectrum analysis also can be obtained by computer analysis.

The data acquisition system of the this study consists of analog instruments as well as a digital system (Figure 3.3). The analog system includes an integrator, DC and RMS voltmeters and a spectrum analyzer. When the digital system is employed, the analog signal from the tracker is first converted into digital signal through 16 bit A/D converter and stored in the microprocessors's user memory. The data is then stored on the floppy disk for further analysis. Software is written for translating the acquired data into the decimal scale.

The computer system of this study is an Apple II microcomputer with one 5.5 inch disk driver. The mean and rms velocities are displayed on the monitor and the stored data are later transfered to the data reduction computer (IBM 3090) when further statistical analysis is desired. The Doppler signal and the velocity signal are also simultaneously monitored on an oscilloscope. The use of both the analog system and the digital system has unique advantage over the use one of those in that accuracy and validation of the data can be easily assesed, and simultaneous data reading and recording can be performed.

3.5 Data Analysis

As discussed in the previous section, the desirable method in the LDV measurement is to obtain the signal which is continuous in time. When the LDV is used in a large scale wind tunnel, or the speed of interest is sufficiently high, the amount of particle seeding in the flow required for continuous scattering becomes
prohibitively large. In addition, the inherent noise problems in LDV applications make it difficult to obtain a continuous velocity signal. Another limiting factor, usually arising in the statistical measurements, is the storage limit of the currently available data acquisition system. This section describes the mathematical development and calculation methods of statistical parameters, with considerations of the signal drop-out and storage limit.

3.5.1 Mean, RMS, Skewness, and Flatness

The moments of the instantaneous velocities are calculated by statistical analysis. The straight arithmetic averages of the moment calculations provide sufficiently accurate results when the sampled data are uniformly distributed in time. Practically, this uniform distribution is difficult to obtain. The main obstacle to this is the signal drop-out and velocity bias. Therefore, two types of correction methods are introduced, the residence time weighting for the signal drop-out and the velocity bias correction for the velocity bias. In the first method, the individual realization of the \( i_{th} \) sampled data \( V_i \) is weighted by the resident time \( \Delta t \), of the realization \( V_i \). The residence time referred to here is the time which the tracker (sample and hold circuit) holds one valid Doppler signal until a new valid signal is retrieved (see Figure 3.6 (c)). The weighting is automatically achieved by selecting the sampling rate less than the drop-out period. The velocity bias arises due to the proportionality of particle flux through the measurement volume to the instantaneous velocity. This fact can be easily visualized from the mass conservation of particles. This gives rise
to a statistical bias towards higher velocities. Therefore, the correction factor must be the inverse of the velocity.

If \( V_i \) is the instantaneous velocity and \( N \) the number of samples taken, then the corrected forms of the moment calculations are as follows:

\[
\text{Mean} = \frac{\sum_{i=1}^{N} V_i W_i}{\sum_{i=1}^{N} W_i} \\
\text{RMS} = \left[ \frac{\sum_{i=1}^{N} (V_i - \bar{V})^2 W_i}{\sum_{i=1}^{N} W_i} \right]^{\frac{1}{2}} \\
\text{Skewness} = \frac{\sum_{i=1}^{N} (V_i - \bar{V})^3 W_i}{\sigma^3 \sum_{i=1}^{N} W_i} \\
\text{Flatness} = \frac{\sum_{i=1}^{N} (V_i - \bar{V})^4 W_i}{\sigma^4 \sum_{i=1}^{N} W_i}
\]

where \( W_i \) is the velocity bias correction function represented by \( W_i = |V_i|^{-1} \), \( \bar{V} \) is the mean velocity, and \( \sigma \) is the rms velocity (standard deviation). Discrepancies between the corrected and uncorrected averages are quantified to be a maximum 4% in the mean velocity and 7% in the rms velocity for the turbulent fluctuations up to 20%.

One further consideration in regard to the moment calculations is what sample size and sampling rate (samples per second here) provide acceptable accuracy and minimize the computational effort. The optimization of the sample size and the sampling rate is essential due to the storage limit of current data acquisition system. Yanta and Smith[48] provided the approximate solution to the question of how many data are necessary to obtain good statistical parameters. Their analysis is based on the assumption that the turbulence is Gaussian (i.e. isotropic turbulence).
One interesting result of their analysis is that the number of data points required is dependent upon the local flow conditions, i.e. the local turbulence intensity. For example, with 20% turbulence intensity, more than 1600 samples are required for 95% confidence limit and less than 1% error in the mean value. A similar analysis by Bates and Hughes[49] showed that the mean, rms, skewness and flatness are mainly dependent on the sample size and weakly dependent on the sampling rate. The analysis of this study on sample size and sampling rate requirements is based on these two previous studies, but is quantified by preliminary tests.

The effects of sample size on the mean, rms, skewness and flatness values are shown in Figure 3.7 for R = 2 at one specific point (x/D = 4, y/D = 6 and z/D = 0). The sampling rate is fixed at 2 KHz for all quantities. As can be seen from the figure, no appreciable variations appear in the mean velocity and turbulence intensity curves. The variation of the mean velocity as N increases from 200 to 5200 is not noticeable and the corresponding variation of turbulence intensity is at most 0.37% (rms variation about the average turbulence intensity from N = 200 to 5200). A sample size N = 10^3 is used for the subsequent calculations of mean velocities and turbulence intensities. The errors are expected to be less than 5% for both quantities.

On the other hand, the skewness and the flatness factors only tend to converge with a sufficiently large sample size (N > 3000), and the scatter in both quantities decreases as the sample size increases. In view of this result, the sample size, N =
2560, which is used for the skewness and flatness factors presented in Chapter 5, may not be sufficiently large to obtain highly accurate results. But, recalling the limited ability of the data acquisition system, a qualitative and some quantitative information on the turbulent structure would be provided from these measurements.

3.5.2 Reynolds Stress

Since the LDV system is a one-component measurement, the Reynolds stress is calculated by averaging the projections of the velocity fluctuations in a plane[6,48]. If the axis of LDA Optics rotates, for example in the x-y plane, to 0, +α, and -α degrees with respect to the x-axis at a point, the three components of the Reynolds stresses can be expressed as

\[
\overline{u^2} = \overline{u_0^2} \tag{3.10}
\]

\[
\overline{v^2} = \left[ \frac{\overline{u_{+\alpha}^2} + \overline{u_{-\alpha}^2}}{\sin^2(+\alpha) + \sin^2(-\alpha)} - \overline{u_0^2} \frac{\cos^2(+\alpha) + \cos^2(-\alpha)}{\sin^2(+\alpha) + \sin^2(-\alpha)} \right] \tag{3.11}
\]

\[
\overline{uv} = \frac{1}{2} \left[ \frac{\overline{u_{+\alpha}^2} - \overline{u_{-\alpha}^2}}{\cos(\alpha) \sin(\alpha) + \cos(-\alpha) \sin(-\alpha)} \right] \tag{3.12}
\]

Other components of the Reynolds stress can be obtained by projecting the instantaneous velocity components in their planes.

3.5.3 Correlations, Length Scales and Power Spectral Densities

The calculations are executed on the IBM 3090 main frame computer after data transfer has been completed. One of the primary motivations for these statistical
Figure 3.7: Effects of sample size on statistical parameters (sampling rate = 2 KHz)
measurements is the acquisition of turbulent scale information. A method is devised
to overcome obstacles due to limited storage space available and the existence of
periods of the signal drop-out.

The autocorrelation functions are first calculated. The autocorrelation function
of the longitudinal velocity component at a delay time, $\tau$, is customarily defined as

$$R(\tau) = \frac{uu^*}{u^21/2u^*21/2}$$  \hspace{1cm} (3.13)

where $u = u(x, y, z, t)$ and $u^* = u(x, y, z, t + \tau)$. The overbar in this equation
denotes the time average. The autocorrelation function can be evaluated by computer
analysis for each delay time, $\tau = m\Delta t$ with $m = 0, \pm 1, \pm 2, ..., \pm M$, using the data
samples of $u(t)$ corresponding to $t = n\Delta t$ with $n = 0, \pm 1, \pm 2, ..., N$
or

$$R(m\Delta t) = \frac{1}{N - 2M} \sum_{n=M}^{N-M-1} u(n\Delta t)u((n + m)\Delta t) \text{ for } |m| \leq M$$  \hspace{1cm} (3.14)

where $N$ is the total number of velocity samples and the time delay is $\tau = m\Delta t$.

In general, the sample time, $(N - 1)\Delta t$, must be at least an order of magnitude
greater than both the longest time scale of the flow field and the maximum delay
time, $\tau_{\text{max}} = (M - 1)\Delta t$. The estimated error is proportional to $N^{-1/2}$ as $N$ becomes
large. However, for $N = IM$, and $I \geq 3$, there exists an efficient algorithm to calculate
the autocorrelation function based on a Fast Fourier Transform (FFT) analysis[50].

The longitudinal integral scale is a convenient measure of the linear extent of
the region within which velocities are appreciably correlated. Mathematically[51]

$$L = \int_{0}^{\infty} f(r)dr$$  \hspace{1cm} (3.15)
where \( f(r) \) is the longitudinal velocity correlation coefficient. When the longitudinal correlation coefficient cannot be directly obtained by a one-component measurement system, the integral length scale is conventionally calculated using the Taylor's hypothesis[52,53]. By rewriting the Eq. 3.15 in terms of the autocorrelation coefficient, \( R(\tau) \), we obtain

\[
L = U_c \int_0^\infty R(\tau) d\tau
\]  

(3.16)

where \( U_c \) is the local mean velocity. Typically the upper limit is chosen at the value of \( \tau \) where \( R(\tau) \) first crosses the \( \tau \) axis.

The Taylor microscale is graphically obtained from the longitudinal correlation coefficient by fitting a parabola near the origin of the coefficient. The Taylor microscale approximately represents the eddy size where dissipation of turbulent kinetic energy is most effective.

By expanding \( f(r) \) in a Taylor series, and by taking into account the symmetry of \( f(r) \) with respect to \( r \), the behaviour of \( f(r) \) in the neighborhood of the origin is expressed as

\[
f(r) = 1 + \frac{1}{2} r^2 \left[ \frac{\partial^2 f}{\partial r^2} \right]_{r=0} + 0(r^4)
\]  

(3.17)

It is common practice to define a length \( \lambda_T \) for very small value of \( r \) as

\[
f(r) \approx 1 - \frac{r^2}{\lambda_T^2}
\]  

(3.18)

The length scale \( \lambda_T \) is called the Taylor microscale. The combination of Eq. 3.17 with Eq. 3.18 yields :

\[
- \frac{1}{\lambda_T^2} = \frac{1}{2} \frac{\partial^2 f(r)}{\partial r^2} \bigg|_{r=0}
\]  

(3.19)
or in terms of \( R(\tau) \)

\[
- \frac{1}{\lambda_f^2} = \frac{U_c}{2} \frac{\partial^2 R(\tau)}{\partial \tau^2} \bigg|_{\tau=0}
\]  

(3.20)

Experimentally, the Taylor microscale is obtained from this equation by numerically fitting the parabola near the origin of the curve.

The power spectral density function of time historic records representing a stationary random process can be defined by the Fourier transform of the correlation function. The Fourier transform of the autocorrelation function can be written as

\[
S(f) = \frac{1}{2\pi} \int_{-\infty}^{+\infty} R(\tau) e^{-j2\pi f \tau} d\tau
\]

(3.21)

Then, \( S(f) \) is called the auto-spectral density function. The auto-spectral density function is accomplished via a Digital Fourier Transform (DFT) originally developed by Cooley and Tukey[54]. The computer algorithm computing the DFT is available from the IMSL (International Mathematics and Scientific Language) subroutine code. This algorithm is especially applicable in cases of poor signal-to-noise ratio and periods of signal drop-out, such as is common in sparsely seeded airflows. The number of calculations required is significantly reduced when the input number of autocorrelation time steps \( M \) is represented by \( M = 2^P \) where \( P \) is any integer from \( M^2 \) to \( M \log M \). The delay time step \( \Delta \tau \) and the maximum time delay \( \tau_{\text{max}} \) are related to the maximum frequency \( f_{\text{max}} \) (called the Nyquist cut-off frequency) and the frequency interval \( \Delta f \):

\[
\Delta \tau = \frac{1}{2f_{\text{max}}} \quad \text{and} \quad \tau_{\text{max}} = \frac{1}{2\Delta f}
\]

(3.22)
In the present study, the autocorrelation functions are first calculated for a number of time steps $M$ equal to 256 with the same delay time interval $\Delta \tau$ as the velocity sample time interval $\Delta t$ ($\Delta \tau = \Delta t = 0.2$ msec).

Examples of the autocorrelation functions using the two different approaches, the conventional averaging technique (Eq. 3.14) and the direct transform method[50], are shown in Figure 3.8. Results are presented for $I = 10$ and 40 ($I = N/M$). Note that the correlation curves become more closely aligned as $I$ increases.

Of principal interest in this investigation, preliminary tests of length scales dependent on the sample size and on the calculation methods of autocorrelation functions are made. A comparison of the integral and Taylor microscales is presented in Figure 3.9 for $I$ from 3 to 50. Several observations are appropriate. The direct transform method yields length scale values which are consistently less than those from the averaging approach. This is true for both the integral and Taylor microscales. Secondly, there is considerably more scatter in the computational results for the integral scale than for the microscale. One explanation for this second observation may arise in the graphical calculation of the Taylor microscale. Although a more accurate value of the Taylor microscale can be obtained with the use of a more fine scale of the delay time, a certain amount of error is included in the Taylor microscale calculations. This may overwhelm the actual variations of the Taylor microscales between two different methods. Thirdly, the direct transform method yields a more accurate integral length scale than the averaging method when a smaller number of
Figure 3.8: Comparison of autocorrelation functions between conventional average and FFT methods.
samples is used (I \leq 20).

The length scale measurements presented in Chapter 5 are obtained by the transform method with the velocity sample N=2560 (I=10). With the current data acquisition system it is difficult to extend the sample size further due to the limited storage space on the disk. Nevertheless, the estimated errors of the length scale data using N=2560 (corresponds to I=10) are expected to be less than 5%, compared to the length scale value using I=50.
Figure 3.9: Comparison of length scales between conventional average and FFT methods for various sample sizes.
Chapter 4

Computational Approach

Recently, computational fluid mechanics has received significant interest due mainly to improved numerical techniques and increased computer efficiency. As a result, a number of turbulence modeling techniques together with efficient numerical solution methods (finite-difference or finite-element) have become accepted as tools analyzing many engineering problems. However, their application to complex flows is still only a practical proposition when there are adequate flow field measurements that can provide test cases.

This chapter describes a computational method applied to the present flow problem. The method is based on the solution of the steady, three-dimensional elliptic forms of the Reynolds time-averaged equations. The Reynolds stresses appearing in the time-averaged equations are calculated by the two-equation \( k-\varepsilon \) model of turbulence.
4.1 Mathematical Model

The flow configuration is such that appreciable variations in the transported quantities such as velocity, temperature, and species arise in all three coordinate directions. It is thus necessary to solve the fully three-dimensional forms of the partial differential conservation equations for mass, momentum and energy or species concentration. Most current turbulence models rely on the solution of the so-called Reynolds time-averaged equations in which the dependent variables are decomposed into time-averaged and fluctuating components.

In Cartesian tensor notation, the time-averaged equations of continuity, momentum, and species concentration describing a steady, three-dimensional flow may be written as

\[
\frac{\partial}{\partial x_i} (\rho U_i) = 0 \tag{4.1}
\]

\[
\frac{\partial}{\partial x_j} (\rho U_j U_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (-\rho \bar{u}_i \bar{u}_j) \tag{4.2}
\]

\[
\frac{\partial}{\partial x_j} (\rho U_j C) = \frac{\partial}{\partial x_j} (-\rho \bar{u}_j \bar{c}) \tag{4.3}
\]

where \( \rho \) is the density, \( p \) is the static pressure, \( u \) and \( c \) are the fluctuating components of velocity and concentration, respectively, and \( U \) and \( C \) are the corresponding time-mean values. In the flow under consideration, molecular diffusion effects are very small compared to turbulent ones, and thus all laminar diffusion coefficients are neglected.

The time-averaged equations are exact, since no assumptions have been intro-
duced in their derivation. However, they cannot be solved in this form, because the new stress and flux terms ($-\rho \bar{u}_i \bar{u}_j$ and $-\rho \bar{u}_j \bar{c}$) arising from the turbulent motion become additional unknowns. Accordingly, additional assumptions regarding the relationship between the turbulent stresses and fluxes and the time mean variables must be incorporated into the turbulence model. Compilations and applications of various turbulence modeling techniques may be found in references [7,8,60].

In the $k-\epsilon$ model, the Reynolds stresses are related to mean strain rate via the Boussinesq eddy viscosity concept:

$$-\rho \bar{u}_i \bar{u}_j = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho \delta_{ij} k$$  \hspace{1cm} (4.4)

where $\mu_t$ is the turbulent or eddy viscosity and $k$ ($= \bar{u}_i \bar{u}_i / 2$) is the turbulent kinetic energy, and $\delta_{ij}$ is the Kronecker delta. The term $\frac{2}{3} \rho k$ can be thought of as the additional pressure resulting from turbulent motion. The heat or mass flux term is obtained using the Reynolds analogy between momentum transport and energy or mass transport:

$$-\rho \bar{u}_j \bar{c} = \frac{\mu_t}{\sigma_t} \frac{\partial C}{\partial x_j}$$ \hspace{1cm} (4.5)

where $\sigma_t$ is the turbulent Prandtl or Schmidt number. The $\sigma_t$ is the ratio of turbulent diffusion coefficient of heat or mass transport to the corresponding momentum transport.

The system is not closed until an expression of the eddy viscosity $\mu_t$ is specified. Kolmogorov [56] suggested in the 1940s that the eddy viscosity can be evaluated as

$$\mu_t = C_u \rho L k^{1/2}$$  \hspace{1cm} (4.6)
where $C_\mu$ is an empirical constant and $L$ is a characteristic length scale. The flow situation under consideration is such that an algebraic expression prescribing the length scale, as in the mixing length model or its modified version, is not adequate to properly simulate the complexities of the flow structure. Although a transport equation can be derived for a length scale, the equation is difficult to model and interpret physically. The proven applicability of the two-equation $k-\epsilon$ model stems from the successful choice of length scale as $L = C_\mu k^{3/2}/\epsilon$. The transport equations for the kinetic energy $k$ and its dissipation rate $\epsilon$ are derived from the Navier–Stokes equations.

The $k-\epsilon$ model has been shown to provide appreciably better universality than lower order models, whereas the higher order closure schemes (the Reynolds stress model) appear at the moment to be insufficiently well developed to prove superior even for two-dimensional flows. It is the purpose of the computational portion of this study to apply and test the $k-\epsilon$ model to the prediction of the present complex three-dimensional situation. The standard version of the $k-\epsilon$ model proposed by Launder and Spalding[57] is used for the calculation procedure.

After a few mathematical manipulations, the governing equations may be written in the following general forms:

$$\frac{\partial}{\partial x_j}(\rho U_j \Phi) = \frac{\partial}{\partial x_j}(\Gamma_\Phi \frac{\partial \Phi}{\partial x_j}) + S_\Phi \quad (4.7)$$

Equations for continuity, momentum, species concentration, turbulent kinetic energy, and dissipation rate of turbulent kinetic energy are presented in Table 4.1,
Table 4.1: The governing equations

<table>
<thead>
<tr>
<th>$\Phi$</th>
<th>$\Gamma_\Phi$</th>
<th>$S_\Phi$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>$U_{i, i = 1, 2, 3}$</td>
<td>$\mu_t$</td>
<td>$-\frac{\partial P}{\partial z_1} + \frac{\partial}{\partial z_2} (\mu_t \frac{\partial U_1}{\partial z_1})$</td>
</tr>
<tr>
<td>$C$</td>
<td>$\mu_t / \sigma_t$</td>
<td>0</td>
</tr>
<tr>
<td>$k$</td>
<td>$\mu_t / \sigma_k$</td>
<td>$G - \rho \epsilon$</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>$\mu_t / \sigma_\epsilon$</td>
<td>$(\epsilon / k)(c_1 G - c_2 \rho \epsilon)$</td>
</tr>
</tbody>
</table>

where $P = p + \frac{2}{3} \rho k$

$$G = \mu_t \left( \frac{\partial U_1}{\partial z_1} + \frac{\partial U_i}{\partial z_i} \right) \frac{\partial U_2}{\partial z_2}$$

and $\mu_t = C_\mu \rho k^2 / \epsilon$

$C_\mu = 0.09, \sigma_t = 0.9, c_1 = 1.44$
$c_2 = 1.92, \sigma_k = 1.0, \sigma_\epsilon = 1.3$

in terms of a general dependent variable $\Phi$, a diffusion coefficient $\Gamma_\Phi$ and a source term $S_\Phi$.

### 4.2 Solution Procedure

The flow geometry under study and coordinate system is shown in Figure 2.1. The flow is symmetric about a vertical plane passing through the center of the jet. The calculations are therefore performed for a rectangular domain, one side of which corresponds to this symmetric plane. Computational results are obtained using a revision of the SIMPLER (Semi-Implicit Methods for Pressure-Linked Equations) algorithm [58]. The detailed description of the algorithm is available [59], and thus only the important features of the procedure are described.

The first step of the solution procedure is the derivation of the finite-difference
forms of governing equations. The finite-difference equations are first formulated by integrating the time-averaged equations over a small control volume surrounding each grid point, along with suitable assumptions about the distribution of the dependent variables between grid points. The combined effects of the convection and diffusion between grid points will be handled by the power-law scheme. The method is formulated in terms of staggered grid arrangements, in which the pressure and other variables are stored in the main grid points and the velocities in staggered locations. A two-dimensional view of staggered grid layout is shown in Figure 4.1 (a). A corresponding three-dimensional grid pattern can be easily visualized in a similar manner. The staggered arrangement is necessary to avoid checkerboard pressure and velocity fields.

As a result, the general form of the finite-difference equations for scalar variables derived at the main grid point P may be written as

$$a_P \Phi_P = a_E \Phi_E + a_W \Phi_W + a_N \Phi_N + a_S \Phi_S + a_T \Phi_T + a_B \Phi_B + b$$

(4.8)

with

$$a_E = D_e A(|P_e|) + [F_e, 0]$$

(4.9)

$$a_W = D_w A(|P_w|) + [F_w, 0]$$

(4.10)

$$a_N = D_n A(|P_n|) + [F_n, 0]$$

(4.11)

$$a_S = D_s A(|P_s|) + [F_s, 0]$$

(4.12)

$$a_T = D_t A(|P_t|) + [F_t, 0]$$

(4.13)
\[ a_B = D_b A(|P_b|) + [[ F_b, 0]], \]  
\[ b = S_c \Delta x \Delta y \Delta z, \]  
\[ a_P = a_E + a_W + a_N + a_S + a_T + a_B - S_P \Delta x \Delta y \Delta z \]  

where \( \Phi_p \) represents the general dependent variables, \( a \)'s are the coefficients resulting from the combined convection and diffusion effects, and \( b \) is the source term containing all terms except the convection and diffusion terms. The upper case subscripts E, W, N, S, T and B refer to the neighborhood grid points around the main grid point \( P \), named by east, west, north, south, top and bottom neighbor, respectively. The lower case subscripts are the corresponding control volume faces. The double bracket \([a,b]\) is a special notation to denote the greater of \( a \) and \( b \).

The Peclet number \( P_e \), which is the ratio of convection to diffusion rate, is given by
\[ P_e = \frac{F_e}{D_e} \]  

where \( F_e \) is the flow rate \((\rho U)_e A_e \) at the control volume face \( e \) with an area \( A_e \), and \( D_e \) is the diffusion conductance given by
\[ D_e = \frac{\Gamma A_e}{(\Delta x)_e} \]  

where \((\Delta x)_e\) is the control volume distance associated with the control volume face \( e \) (see Figure 4.1 (b)).

The term \( A(|P|) \) in Eq. 4.9–Eq. 4.14 is the general notation categorizing various convection–diffusion schemes, such as upwind, central, hybrid, power–law, and
Figure 4.1: Grid arrangement; (a) two-dimensional view of staggered grid layout, (b) one-dimensional grid clustered for grid point P.
exponential scheme. In the power-law scheme, the \( A(|P|) \) may be expressed as

\[
A(|P_e|) = [0, (1 - 0.1|P_e|)^5]
\]  

(4.19)

Here again the double bracket \([ , ]\) is a notation which takes the larger value of the two in the bracket. The power-law scheme provides a more realistic distribution of a convection-diffusion profile between two grid points than any other schemes except the exponential scheme. The exponential scheme, however, requires more computational time than any other convection-diffusion scheme. Notice further the source terms \( S_C \) and \( S_P \). The source term \( S_\star \), originally defined in the derivation of the general form of the governing equation (Eq. 4.7), is further divided into two parts such as \( S_\star = S_C + S_P \Phi_P \). The \( S_C \) is the constant part of \( S_\star \), while \( S_P \) is the coefficient of \( \Phi_P \). The coefficient term \( S_P \) must be always less than or equal to zero to satisfy the stability requirement of the numerical scheme.

The finite-difference equations for the momentum equations are derived in a manner analogous to those of scalar variables, but they are formulated at the staggered locations. Special attention, however, is required for the pressure terms. Note that there is no exact equation expressing the pressure term. Focusing on \( U \) only and explicitly separating the pressure gradient from the source term, \( b \), the resulting finite-difference equation of \( U \) derived at the control volume face \( e \) can be written as

\[
a_e U_e = \sum a_{nb} U_{nb} + b + (p_P - p_E) A_e
\]

(4.20)

where the summation term, \( \sum a_{nb} U_{nb} \), denotes the sum of the neighbor terms of \( U_e \).
The momentum equation can be solved only when the pressure field is provided. Unless the correct pressure field is employed, the resulting velocity field will not satisfy the continuity equation. This problem is handled by successive iterations with an appropriate correction to the velocity field at each iteration step. A few steps of this iteration scheme will be developed.

The imperfect velocity field based on a trial pressure field (will be denoted by \( p^* \)) results from the solution of the following equation:

\[
a_e U_e^* = \sum a_{nb} U_{nb}^* + b + (p_P^* - p_E^*) A_e
\]  

(4.21)

The problem is now to find an appropriate correction formula to this imperfect velocity field such that the imperfect velocities in successive iterations converge, and this solution will ultimately satisfy the continuity equation. Suppose that the trial pressure field can be corrected by a pressure correction \( p' \) as

\[
p = p^* + p'
\]  

(4.22)

and next the velocity field is corrected by a velocity correction \( U_e' \):

\[
U_e = U_e^* + U_e'
\]  

(4.23)

Then, the relation between the pressure correction and the velocity correction can be obtained by subtracting Eq. 4.21 from Eq. 4.20, together with the use of the relations in Eq. 4.22 and Eq. 4.23:

\[
a_e U_e' = \sum a_{nb} U_{nb}' + (p'_P - p'_E) A_e
\]  

(4.24)
The final form of the velocity correction formula is obtained by temporarily dropping the term, $\sum a_{nb}U_{nb}^*$, from this equation and by replacing $U_{e}^*$ using Eq. 4.23:

$$U_{e} = U_{e}^* + d_{e}(p_{p}^* - p_{p}^{'})$$

(4.25)

where $d_{e} = A_{e}/a_{e}$. Thought not presented here, the correction formula for the velocity components in other directions can be obtained in a similar manner. Now, having an expression for solving $p'$, the system of solution scheme is essentially complete.

The $p'$ equation is derived by substituting velocity corrections for all velocity components into the continuity equation. The $p'$ equation is called the pressure correction equation and given by

$$\alpha_{P}p'_{p} = \sum a_{nb}p'_{nb} + b$$

(4.26)

where

$$b = (\rho U^* A)_{w} - (\rho U^* A)_{e} + (\rho V^* A)_{s} - (\rho V^* A)_{n} + (\rho W^* A)_{b} - (\rho W^* A)_{t}$$

(4.27)

If $b$ in this equation is zero, the starred velocities satisfy the continuity equation and iteration terminates. The term $b$ thus represents a “mass source”, indicating the extent to which the velocity field does not satisfy mass conservation considerations at a particular iteration stage.

The solution procedures introduced up to this point are the basic algorithm of the SIMPLE methods. In an attempt to improve the rate of convergence, a revised version of SIMPLE (called SIMPLER) is more effectively worked out[59]. SIMPLER
continues to use the pressure correction equation, but employs a separate pressure
equation to more effectively predict the pressure field than the SIMPLE method.
The motivation for this arises from the rather exaggerated velocity corrections (Eq.
4.25), which are obtained by omitting the term $\sum a_{nb} U'_{nb}$ when the $p'$ equation is
derived.

Introducing a velocity $\hat{U}_e$ (called the "pseudo velocity") by

$$\hat{U}_e = \frac{\sum a_{nb} U_{nb} + b}{a_e}$$  \hspace{1cm} (4.28)

the momentum equation for $U_e$ (Eq. 4.20) can be rewritten as

$$U_e = \hat{U}_e + d_e (p_P - p_E)$$  \hspace{1cm} (4.29)

where $d_e = A_e/a_e$. In a manner analogous to the development of the pressure
correction equation, the final form of the pressure equation may be written:

$$a_P p_P = \sum a_{nb} p_{nb} + b$$  \hspace{1cm} (4.30)

where

$$b = (\rho \hat{U} A)_w - (\rho \hat{U} A)_e + (\rho \hat{V} A)_s - (\rho \hat{V} A)_n + (\rho \hat{W} A)_b - (\rho \hat{W} A)_t$$  \hspace{1cm} (4.31)

The pressure field from this equation is a direct consequence of the given velocity
field, and thus the convergence is faster than any other solution procedure containing
only the pressure correction equation.

All of the equations needed for obtaining velocities, pressure and other scalar
variables have been developed. The iteration procedures are now summarized. The
finite-difference equations are solved by line-by-line iterations in which all the variables along a line are simultaneously solved with temporarily fixed variables lying off the line. The pressure field is first calculated from Eq. 4.30, using an initially guessed or given velocity field from the previous iteration step. Then, an intermediate velocity field (starred velocities) is obtained from the momentum equation (Eq. 4.21). This velocity field is once again connected to the continuity equation to obtain the pressure correction equation (Eq. 4.26). This velocity field, in general, does not satisfy the continuity equation. Therefore, corrections to the velocity field (Eq. 4.23) are made until the latter solution satisfies both the momentum equations and the continuity equation.

4.3 Boundary Conditions

The governing equations appropriate to the present flow configuration require boundary conditions for the dependent variables on all of the boundary surfaces of the solution domain. The boundaries are the inlet plane of the cross stream, the downstream plane, one symmetric plane passing through the center of the jet, one side wall, and two confining plates. Whenever the velocity is specified at a boundary, the boundary condition for the pressure is not necessary, because only the relative magnitude of pressure force plays a role in the momentum equations.

The upstream boundary conditions are prescribed, as far as possible, from the experimental measurements. These data include the $x$-component mean velocity, the turbulent kinetic energy, and the length scale. At the symmetric plane, Neu-
mann boundary conditions are applied, i.e. the normal gradients of all variables except the normal velocity component are taken to be zero. The normal velocity is set to zero (W=0).

At the downstream plane the gradients of all dependent variables in the x-direction are equal to zero. These conditions may not correspond to those prevailing in the real flow. The conditions are satisfactory only when the fully developed profile assumptions of the variables are valid. This requirement locates the boundary unnecessarily far downstream (for example, x/D>100 with R=6). Practically, the boundary location is adjusted such that the specified boundary conditions, which in a certain range differ from the fully-developed conditions, have negligible influence in the region of interest. For the variations of all dependent variables less than 1%, the conditions are achieved by computational trials, approximately at x/D=24 for R=2, x/D=32 for R=4, and x/D=48 for R=6.

The boundary conditions on the walls require special considerations. This is primarily due to the significant effects of molecular viscosity. Note that the two-equation model of turbulence previously introduced has neglected the molecular viscosity. The neglect of molecular viscosity is valid only in the fully turbulent regions. A two-layer model of the wall function method proposed by Launder and Spalding[60] is utilized in the present computational study. Usual no-slip conditions are still valid for the convection fluxes. For the two velocity components parallel to
the wall, the diffusion fluxes are patched onto the wall law profiles:

\[ \mu_{eff} = \mu \quad \text{if} \quad y^+ < 11.5 \quad (4.32) \]

and \[ \mu_{eff} = \frac{\mu y^+}{(1/\kappa) \ln(E y^+)} \quad \text{if} \quad y^+ \geq 11.5 \quad (4.33) \]

where \( \kappa \) is the Von-Karman constant (=0.41) and \( E \) is another constant with a value of 9.0. The \( y^+ \) is the normalized distance of the first internal grid point from the wall, defined by

\[ y^+ = \rho k^{1/2} C_{\mu}^{1/4} y_I \quad (4.34) \]

where \( y_I \) is the actual distance of the grid from the wall. Because of the considerable variations of turbulent quantities near the wall, the mean generation rate \( \tilde{G} \) and the mean dissipation rate \( \epsilon \) of the turbulent kinetic energy appearing in the governing turbulent kinetic equation (Table 4.1) are evaluated using the profile assumptions of the turbulent quantities near the wall cell.

All of the dependent variables also must be specified at the jet exit plane. The half-circular nozzle, however, is modified by the rectangular cells in the Cartesian coordinate system (Figure 4.2). The cells' surface areas are adjusted such that the specified jet velocity provides the correct mass flux through the surface. In order to investigate the influence of the jet boundary conditions on the computed results, two different profiles are tested as the jet field's initial conditions for \( R=2 \) and 4. The first is the uniform profile (top hat shape in Figure 4.3), characterizing the flow exiting a contraction/nozzle arrangement. The second profile maintains the same momentum flux, but is skewed downstream.
Figure 4.2: Modification of the half-circular nozzle by rectangular cells
Figure 4.3: Mean velocity and mean temperature profiles at the jet exit plane for $R = 2$
The velocity and species concentration profiles used for the second condition are from the measurements by Andrepoulos[18,19]. His measurements revealed that the jet stream at the exit plane is distorted due to the pressure gradients across the plane and the distortion increases as the velocity ratio $R$ decreases. Figure 4.4 shows the longitudinal mean velocity profiles non-dimensionalized with the free-stream value, $U/U_o$, plotted versus vertical distance. The maximum deviation of $U/U_o$ between two conditions at $x/D=0.0$ is at most 6% for both velocity ratios. After $x/D > 2$, there are essentially negligible differences between profiles with two different boundary conditions. Though not depicted here, the same observations are correct for the profiles of the scalar concentration field.

One conclusion reached from these observations is that only the total momentum flux at the jet exit plane (not its profiles) is important for the development of the flow field downstream. Physically, this means that near the jet discharge the flow field is not significantly influenced by viscous effects, so that the viscous diffusion due to the velocity gradients at the jet exit plane may be neglected. The first boundary condition is chosen for the subsequent calculations because it is the simplest and most generally used in computational situations. Particularly for the flow situation under study, the realistic profile of the second condition is not expected to provide appreciably better results over the first, simpler one.

The boundary conditions described above are tabulated in Table 4.2. In this table, $W(T/B)$ denotes the wall function applied both to the top and bottom walls,
Figure 4.4: Comparison of mean velocity, $U/U_0$, between two different jet exit boundary conditions in the plane of symmetry; (a) $R = 2$, (b) $R = 4$
Table 4.2: A summary of boundary conditions

<table>
<thead>
<tr>
<th></th>
<th>Upstream</th>
<th>Downstream</th>
<th>Symmetry</th>
<th>Jet exit</th>
<th>Walls</th>
</tr>
</thead>
<tbody>
<tr>
<td>$U$</td>
<td>$U_o$</td>
<td>$\partial U/\partial x = 0$</td>
<td>$\partial U/\partial z = 0$</td>
<td>0</td>
<td>$W(S), W(T/B)$</td>
</tr>
<tr>
<td>$V$</td>
<td>0</td>
<td>$\partial V/\partial x = 0$</td>
<td>$\partial V/\partial z = 0$</td>
<td>0</td>
<td>$W(S)$</td>
</tr>
<tr>
<td>$W$</td>
<td>0</td>
<td>$\partial W/\partial x = 0$</td>
<td>0</td>
<td>$V_j$</td>
<td>$W(T/B)$</td>
</tr>
<tr>
<td>$C$</td>
<td>0</td>
<td>$\partial C/\partial x = 0$</td>
<td>$\partial C/\partial z = 0$</td>
<td>1</td>
<td>$\partial C/\partial N^*$</td>
</tr>
<tr>
<td>$k$</td>
<td>$(3/2)u_o^2$</td>
<td>$\partial k/\partial x = 0$</td>
<td>$\partial k/\partial z = 0$</td>
<td>$(1/2)(v_j^2)$</td>
<td>$W(S), W(T/B)$</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>$k^{3/2}/(0.165H)$</td>
<td>$\partial \epsilon/\partial x = 0$</td>
<td>$\partial \epsilon/\partial z = 0$</td>
<td>$k^{3/2}/(0.5D)$</td>
<td>$(C_{\mu}^{3/4}k^{3/2})/(\kappa y_1)$ **</td>
</tr>
</tbody>
</table>

* Impermeable wall boundary condition
** Local equilibrium profile specified on the grids nearest the wall

and $W(S)$ applied to the side wall. At all grid points near the wall, the local equilibrium values of the turbulent quantities replace the dissipation rate $\epsilon$, instead of solving it from the transport equation. The equilibrium profile is given in the bottom and right corner in Table 4.2. The isotropic assumption is used for the boundary condition of the turbulent kinetic energy at the upstream boundary plane.

### 4.4 Computational Details

The computations are executed on an IBM 3090 at Louisiana State University. In order to minimize the number of grid points to be used, non-uniform spacing is used so that grid nodes could be clustered where rapid variations of dependent variables are expected. This means that fine grid spacing is used near the jet discharge, and an increasingly larger grid spacing is used away from the discharge.
hole in all three coordinate directions. The location of the grid line nearest the wall is adjusted such that all grid points on the line occur in the fully turbulent region \(11 < y^+ < 300\) where the logarithmic wall law profiles are valid.

At each iteration it is necessary to employ underrelaxation when solving the algebraic, finite-difference equations (i.e. \(\Phi = \lambda \Phi_{\text{new}} + (1 - \lambda) \Phi_{\text{old}}\)). The number of iterations and the stability or divergence of the solutions are directly affected by the value of the underrelaxation factor \(\lambda\). Typical values of \(\lambda\) used are \(\lambda = 0.2\) to 0.4 for the velocity components and \(\lambda = 0.5\) for all scalar variables. The iteration terminates when the normalized sum of the mass source with respect to inflow mass flux \((b\) in Eq. 4.30), which represents the deviation of the velocity field from the mass conservation, is less than \(10^{-5}\) and the variations of all dependent variables between two successive iterations are less than 0.1%. A typical calculation using 20 \(\times\) 15 \(\times\) 15 points \((x, y\) and \(z\) direction, respectively) required approximately 250 iterations and 14 minutes of CPU time.

Computer storage limitations in the 3-D calculation necessitate the use of a relatively coarse grid distribution. The actual variations of the dependent variables between grid points in the convection-diffusion problems exhibit an exponential behavior (power-law scheme represents this behavior). A truncated Taylor series in an upwind or central difference scheme fails to be an adequate representation of the exponential behavior except for fine grid size[61]. The power-law scheme adapted in this work provides an acceptable representation of the exponential behavior and
therefore minimizes false diffusion. Deuren[61] provides estimation methods of false diffusion in 3-D calculations. However, for the present work, with the computer storage available at this time, it is difficult to refine the finite-difference grid further. There are, however, indications that the results are grid dependent, as the contour shapes are repeatable with a coarser grid size (15 x 10 x 10 in the x, y and z directions, respectively) but the magnitude of, for example, the x-component of mean velocity may vary up to 10%.
Chapter 5

Results and Discussions

The ultimate objective of this study is the clearer understanding of the jet in a confined cross flow. Primary interest is in the jet impingement process, and thus large values of the velocity ratio are considered, in particular, \( R = 2, 4 \) and \( 6 \). The channel separation is fixed at ten jet diameters for all velocity ratios. The applications of the results will be in V/STOL aerodynamics and the turbine blade cooling problems, as well as other areas.

The characteristics of inflow boundaries are investigated first. The effect of the jet exit profile on the subsequent flow development downstream and upstream boundary is documented. Detailed mappings of the mean and fluctuating velocity components of the flow field downstream are made. The resultant information will improve the understanding of this particularly relevant flow problem and its relationship to the unconfined configuration.

The structured nature of the turbulent flow field is documented from the measurements of statistical properties. This investigation is directed towards a search
for the existence of large scale, deterministic structures in the turbulent flow field. Measurements of autocorrelation functions, power spectra and length scales serve as tools providing such information. The motivation for determining properties such as probability functions, skewness and flatness factors of the instantaneous velocities arises from the non-homogeneous nature of the flow field.

The calculated results are compared to experimentally obtained data. Comparisons made include mean velocities, turbulent stress and dissipation rate of the turbulent kinetic energy.

5.1 Boundary Conditions and Their Effects

The investigation of the jet's initial conditions is of utmost importance. The jet's initial profile has a strong influence on the subsequent flow development. The turbulent free jet has been experimentally investigated in the plane located one diameter above the jet's exit plane (y/D = 1.0). Included are the longitudinal mean velocity and turbulence intensity profiles, autocorrelation functions, auto-spectral densities, Taylor microscales, and integral length scales.

In Fig 5.1, mean velocity and turbulence intensity profiles are presented in the x-z plane at y/D = 1.0 for R=2 and 4. The ratio of the local mean velocity to the centerline mean velocity, \( V/V_j \), is plotted versus the non-dimensionalized lateral coordinate, z/D. The mean velocity profiles exhibit a "top hat" shape, and from these profiles the assumption of the symmetry of the flow field with respect to the jet axis (z/D = 0) is validated. The deviation from the symmetry is less than 1%.
everywhere except the outer edge of the jet for both velocity ratios. The average values of the turbulence intensities are approximately 4.5% for R=2 and 1.5% for R=4.

In Fig. 5.2, autocorrelation functions for R=2 and 4 are presented in the potential core located one jet diameter above the jet exit plane (y/D = 1.0). Note that, for R=2, the autocorrelation function resembles a curve that would be expected in the fully turbulent region, while, for R=4, a damped sinusoidal nature of the autocorrelation curve results. The sinusoidal nature of this curve in case of R=4 indicates the passage of the vortical rings shed from the lip of the nozzle. Clearly, the relatively higher local turbulence intensities for R=2 (Fig. 5.1) have masked out the vortical structures, and thus the autocorrelation curve in this case has been damped out with relatively weaker sinusoidal behavior than the case of R=4.

The auto-spectral densities are shown in Fig. 5.3 at the same location as those of the autocorrelation functions. The spectrum is normalized with the mean square of the jet velocity fluctuations. The peaks in the spectrum for R=4 again indicate the passage of vortical structures, reinforcing the observation made relative to the autocorrelation function. The spectral curve for R=2 does not exhibit a peak.

The integral length scales and Taylor microscales are constructed at y/D = 1.0 and x/D = 0.0 for three lateral positions (Table 5.1). Note that the Taylor microscales are more uniform across the jet cross-section than are the integral scales. This result is consistent with the idea that the turbulent energy generation is primar-
Figure 5.1: Near field jet mean velocity and turbulence intensity profiles at $y/D=1.0$ and $x/D=0.0$
Figure 5.2: Autocorrelation functions of turbulent free jet in the potential core located at $y/D=1.0$
Figure 5.3: Auto-spectral densities of turbulent free jet in the potential core at $y/D=1.0$
Table 5.1: Integral length scales and Taylor microscales for the free jet at y/D = 1.0 and x/D = 0.0

<table>
<thead>
<tr>
<th>z/D</th>
<th>L/D</th>
<th>λ_T/D</th>
<th>L/D</th>
<th>λ_T/D</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>1.65</td>
<td>0.48</td>
<td>2.11</td>
<td>0.64</td>
</tr>
<tr>
<td>0.2</td>
<td>2.25</td>
<td>0.68</td>
<td>3.42</td>
<td>0.95</td>
</tr>
<tr>
<td>0.4</td>
<td>4.54</td>
<td>0.62</td>
<td>6.08</td>
<td>0.75</td>
</tr>
</tbody>
</table>

ily a large scale phenomenon. The integral length scale is an experimental estimate of the largest scale of the turbulent flow field. The Taylor microscale approximately determines the size of the energy dissipation scale.

The effect of the jet flow on the upstream boundary is experimentally investigated. One motivation of this measurement is in the acquisition of boundary values to be used in the computational work. The usual assumed boundary condition requires this boundary plane to be located unnecessarily far upstream. All measurements in this investigation are made at eight jet diameters upstream from the jet center in the plane of symmetry (x/D = -8.0 and z/D = 0.0).

Mean velocity and turbulence intensity profiles of the tunnel flow are shown in Fig. 5.4. Attention is focussed on three cases: (1) no jet flow, (2) jet flow on for R = 2 and (3) jet flow on for R = 4. While the mean velocity profiles are nearly independent of the jet flow condition, the turbulence intensities are significantly amplified by the jet flow, with the average turbulence intensities ranging from 1.1% to 6.8% for R = 2 and from 1.1% to 7.1% for R = 4. This increase in turbulence intensities is a result of the bluff-body type wake formed near the leading edge of the jet.
Figure 5.4: Effects of the jet flow on the upstream boundary at $x/D = -8.0$ and $x/D = 0.0$; (a) mean velocity, (b) turbulence intensity
Figure 5.5: Integral length scale, $L$, and Taylor microscale, $\lambda_T$, for the upstream boundary at $x/D = -0.8$ and $z/D = 0.0$
The integral length and Taylor micro scales at the upstream boundary are presented in Fig. 5.5. The average value of the integral scale, which is used for the length scale in the formulation of the boundary condition of the dissipation rate (Table 4.2), is approximately \( L = 1.65 \) \( D \) (dotted line in Fig. 5.5), with a maximum variation of approximately 35\% for both velocity ratios. The Taylor microscales and integral scales are nearly independent of the value of the velocity ratio \( R \). Variations of the integral length scale are considerably more pronounced than those of the Taylor microscale for both velocity ratios. One explanation for the scatter in these results is that the Taylor microscale is not as strongly dependent as the integral scale on the initial turbulence generated by the jet’s injection. That is, the generation is primarily a large scale phenomenon. The energy cascade process required to transfer energy to the Taylor microscales serves to average out the turbulence generated in the anisotropic, near field.

5.2 Mean Flow Fields

The trajectory of the jet in a confined cross flow is of paramount importance. The jet trajectory defined here is the locus of the maximum mean velocity in the plane of symmetry. The calculated jet trajectories for \( R=2, 4 \) and 6 are plotted in Fig. 5.6. Here comparison is made to experimentally obtained jet trajectories by Ramsey and Goldstein[11] and Keffer and Bains[9] for unconfined cross flows. The calculated trajectories are less steeply turned downstream than the unconfined experimental data exhibits. Thus, the “apparent” value of \( R \) is higher for the com-
putational, confined case. Alternatively, the calculated results may underestimate the influence of viscosity. A less viscous fluid would permit the jet to penetrate farther into the outer stream. A false diffusion of the computational model due to the relatively coarse grids used would be an additional reason for the difference. Note that the agreement improves near the jet's exit plane where relatively fine grids are used.

A similarity plot of the computed trajectories for the three different values of the velocity ratio is presented in Fig. 5.7. The similarity function used is [33]:

\[
\frac{y}{D} = aR^b\left(\frac{x}{D}\right)^c
\]

(5.1)

where \(a, b\) and \(c\) are empirical constants with the values equal to 0.977, 0.911 and 0.335, respectively. This equation implies that the jet trajectories are dependent only on the velocity ratio. Results show that the agreement among the calculated results for \(R = 2, 4,\) and \(6\) is quite good, and thus a single curve can represent the velocity trajectories for the range of the velocity ratios used in this work. The computed jet trajectories, compared to the experimentally determined curve (Eq. 5.1), reinforce the observations that the computed results again overestimate the effective value of \(R\).

Fig. 5.8 shows isocontour plots of the non-dimensionalized species concentration, \(C/C_J\), in the plane of symmetry. The contours indicate the extent of the penetration of the marked jet particles into the outer stream. The line, \(S\), of maximum maximorum (i.e. maximum of maximums[62]) is constructed for each velocity ratio.
Figure 5.6: Maximum velocity trajectories in the plane of symmetry
Figure 5.7: Similarity profiles of jet trajectories in the plane of symmetry: \( \frac{y}{D} \). 
\( \frac{y}{D} / R^b = a(z/D)^c \) with \( a = 0.977 \), \( b = 0.911 \), and \( c = 0.335 \)
From this line, a qualitative representation of the flow field is achieved including the extent of the jet deflection and the existence of impingement. For larger velocity ratios (in this case, \(R = 4\) and \(6\)), the jet stream shifts dramatically to the top plate, and after impingement a three-dimensional wall jet-type flow develops along the top wall. However, at the lower velocity ratio (\(R = 2\)), impingement does not occur, and negligible effects of the confining surface on the flow field are expected. The trajectory of the maximum concentration, line \(S\), for \(R = 2\) is aligned with the cross stream approximately at \(x/D = 6.0\), after a rapid rise near the jet entrance.

Transverse cross-sections of the scalar isocontours at \(x/D = 1.5, 3.5\) and \(7.5\) for \(R = 2, 4\) and \(6\) are presented in Fig. 5.9. For \(R = 2\), and recalling the limited penetration of the jet at the lower velocity ratio, the importance of the bottom plate is evident. The isocontours also illustrate that the diffusion characteristics of the jet stream strongly depend on the velocity ratio. For the lower velocity ratio (\(R = 2\)), after rapid deflection in the initial region, the jet stream is convected downstream and diffuses out in both the vertical and transverse directions (\(y\) and \(z\)-directions, respectively). For larger velocity ratios (\(R = 4\) and \(6\)), the jet stream directly impinges on the opposite plate and diffuses more rapidly in the side direction (\(z\)-direction). The kidney-shaped cross-section of the jet is clearly seen as the jet develops downstream. The location of the maximum scalar value occurs on the top wall and along the center line after impingement.

In Fig. 5.10, the total mean velocity vectors are plotted in the plane of symmetry
Figure 5.8: Predicted scalar concentration, $C/C_J$, and line of maximum maximum, $S$, in $x$-$y$ plane; (a) $R=2$, (b) $R=4$, (c) $R=6$
Figure 5.9: Predicted scalar field, \( \frac{C}{C_0} \), in y-z plane; (a) \( R = 2 \), (b) \( R = 4 \), (c) \( R = 6 \)
and the results demonstrate the existence of reverse flow immediately behind the jet. The wake-type region behind the jet extends downstream with the increasing velocity ratio. For the case of R=2, the jet stream is deflected rapidly downward near the jet discharge, similar to that of the scalar field (Fig. 5.8). As the velocity ratio increases, significant upward motion continues further downstream.

Fig. 5.11 shows contours of the x-direction mean velocity in y-z planes at two downstream locations, x/D = 0.5 and 7.5, for R=2, 4 and 6. The velocity is non-dimensionalized with the cross stream velocity. In the initial region (x/D = 0.5), the crossflow is accelerated around the edge of the jet and produces velocity maxima similar to the flow around a cylinder. The location of the maxima arises on either side of the symmetric plane and near the bottom wall. The maximum streamwise velocity $U_{max}$ reached is approximately 1.11$U_o$ for R=2, 1.22$U_o$ for R=4, and 1.36$U_o$ for R=6. The behavior of the crossflow evidences the increasing acceleration around the jet fluid as R increases. In the limit as R approaches infinity, the acceleration would be comparable to the inviscid case when $U_{max} = 2U_o$.

Another important observation from Fig. 5.11 is the fact that the value of the mean velocity $U/U_o$ is less than unity inside the jet cross-section. The value $U/U_o$ less than unity indicates that, in the near field region, the initially vertical momentum of the jet is not immediately transferred to the horizontal direction. Note that the jet stream gradually recovers the axial direction velocity as it is convected downstream. The cross stream starts to accelerate the jet stream from the edges of
the jet cross-section after the jet is turned downward, and eventually two streams will exhibit the same axial momentum. Again as the velocity ratio increases, the impingement on the top wall is clearer. Downstream of the impingement the jet flattens against the top wall and rapidly spreads in the side direction (z-direction). Similar behavior to this has been previously observed from the scalar isocontours shown in Fig. 5.9

A different perspective on the interaction of the jet and the confined cross flow for R=2 is demonstrated in Fig 5.12 and Fig 5.13. These plots provide an overview of the velocity field, including the wake behavior, interaction between the jet flow and the cross flow, and the momentum deficit. In Fig. 5.12, the velocity, \( U/U_o \), is plotted for one half of the cross-sectional y-z plane at four downstream locations, \( x/D = 0.0, 1.0, 3.0 \) and \( 6.0 \). Thus, observations can be made with respect to the momentum deficit that results from the jet interaction. Note the acceleration of the mean flow immediately outside the wake region and near the bottom plate (Fig. 5.12 (a)). The wake extends farther in the direction of the jet than in the cross stream direction. The highest shearing rates exist near the jet entrance to the flow. These shear forces create a vortex pair and the region of the vortex pair is bounded to the circumferential edges of the jet cross-section in the kidney shape (will be discussed in detail later). The jet directed dimensionless velocity, \( V/U_o \), is presented in Fig. 5.13 for R=2 at \( x/D = 0.15, 1.5, 3.5 \) and \( 7.5 \). Note that the maximum value of \( V/U_o \) decreases downstream. At \( x/D = 7.5 \), \( V/U_o \) is equal to
Figure 5.10: Total mean velocity vectors in x-y plane; (a) R = 2, (b) R = 4, (c) R = 6
Figure 5.11: Streamwise mean velocity isocontours, $U/U_0$, in y-z plane: (a) $R=2$, (b) $R=4$, (c) $R=6$
approximately 0.5 at the plane of symmetry.

A higher velocity ratio case, $R = 4$, is documented in Fig. 5.14 and Fig. 5.15. In Fig. 5.14 (a)-(d), $U/U_o$ is plotted in the y-z plane for $x/D = 0.0$, $1.0$, $3.0$ and $6.0$. A comparison with the results shown in Fig. 5.12 is informative. For $x/D = 0.0$, the flow field with $R = 4$ is characterized by greater acceleration of the cross flow near the jet entrance as described previously and an increase in the extent of the jet induced wake. Also, near the top surface ($y/D = 10.0$), there exists a secondary wake region. This flow would exhibit essentially a three-dimensional boundary-layer type of flow developing along the top wall. This did not occur for $R = 2$. In Fig. 5.15 (a)-(d), $V/V_o$ is graphed in the y-z plane for $x/D = 0.15$, $1.5$, $3.5$ and $7.5$. Note the rapid acceleration and then deceleration the jet flow exhibits. Both occur at a much higher rate than shown in the $R = 2$ case. The higher velocity gradients indicate the probability of a greater transfer of energy from the mean to the turbulent flow.

As mentioned earlier in Chapter 2, one essential feature of the jet in a crossflow is the production of the counter-rotating vortices. Numerical calculations of this study predict the vortex production (Fig. 5.16). As the jet enters the cross flow, its shape begins to change because of the total pressure force across the jet's cross-section plane. The jet is deformed into a kidney shape and the cross flow creates a pair of vortices behind the jet in much the same way as a flow around a cylinder.

The streamwise component of mean vorticity is calculated from the velocity
Figure 5.12: Isometric view of the interaction of jet and crossflow in x-direction for $R = 2$
Figure 5.13: Isometric view of the interaction of jet and crossflow in y-direction for R=2
Figure 5.14: Isometric view of the interaction of jet and crossflow in x-direction for R=4
Figure 5.15: Isometric view of the interaction of jet and crossflow in $y$-direction for $R=4$
vector matrix in the cross-section planes. Here, the vorticity is defined as follows:

\[ \Omega_z = \frac{\partial(W/U_0)}{\partial(y/D)} - \frac{\partial(V/U_0)}{\partial(z/D)} \]  

(5.2)

Only the vortices in the half-plane extending from the symmetric plane are presented. The opposite half-plane can be visualized with an opposite sign for the vortex strength. The core of the vortex structure in each plane is seen to propagate towards the upper surface as it is convected downstream. The cross-sectional shape of the vortex structure seems to be similar to that of scalar field drawn in same cross-section planes (see Fig. 5.9) except near the bottom wall. It is interesting to note that the vortex structure still exists after the jet impinges the wall.

As seen in Fig. 5.16, the strength of the mean vorticity decreases downstream. Diffusion by turbulent stress is mostly responsible for the reduction. The lateral spreading rate of the vortex field downstream is also directly dependent on diffusion effects. According to Moussa et al.[14], the vortex rings shed from the lip of the jet nozzle are reoriented and bundle up into a counter-rotating vortex pair that are bound to the lee surface of the jet. Andreopoulos and Rodi[16] speculated further about the nature of the vortex production mechanism, and proposed that the development of vortex rings is enhanced by the oncoming vorticity in the boundary layer of the cross stream, which develops along the bottom wall.

However, the major mechanism that generates the vortex pair is the strong shearing force at the interface sheet between the jet and the cross stream. Near the jet discharge, there exists a strong velocity gradient of the streamwise velocity
Figure 5.16: Streamwise mean vorticity contours at two downstream locations in y-z plane; (a) R=2, (b) R=4, (c) R=6
component with respect to the lateral direction, $\partial U/\partial z$. The velocity gradient in conjunction with the viscosity initially creates the y-direction vorticity. This is transferred into the x-direction as the jet is deflected downward.

5.3 Comparison of Calculated Results to Measurements

To establish the credibility of the computational results, the comparison is made to the experimentally obtained results. Two components of mean velocities are compared to experimental results at four downstream locations in the x-y plane (Fig. 5.17 for $R=2$ and Fig. 5.18 for $R=4$). The agreement is generally fair, though some difference is exhibited in the upstream region. The main discrepancy is in the axial direction velocity component $U$, again in the upstream region where the flow exhibits strong anisotropy. The precise reasons for the discrepancies in these results, however, are difficult to identify. Although detailed tests of the grid dependency on the calculation results were not possible due to storage limits in this 3-D calculations, the relatively coarse grids used ($20 \times 15 \times 15$) would be insufficient to resolve the significant variations of variables in all three coordinate directions. Generally, the predictions appear to exhibit diffusion rates smaller than those of the experimental situations: i.e. the jet penetrates further into the cross stream in the computational model. This observation was also made in the comparison of the jet trajectories (see Fig. 5.6).

At $x/D = 2$ for $R=2$, the measured profiles of $U/U_o$ shows a peak approximately
Figure 5.17: Comparison of mean velocities between predictions and measurements in x-y plane for \( R = 2 \); •, measurement, —, prediction.
Figure 5.18: Comparison of mean velocities between predictions and measurements in x-y plane for R = 4; •, measurement, —, prediction
at \( y/D = 5.0 \). This peak is the result of the jet which has been transferred from the vertical to horizontal directions. The relatively low resolution of the calculation methods of this study does not predict the peak. The streamwise component velocity profiles both at \( R = 2 \) and \( R = 4 \) clearly show wake behavior developing between the jet and the bottom plate. The wake region is induced by the backflow of the cross stream into the low pressure region immediately downstream of the jet discharge. The wake region extends downstream but "lifts off" from the lower wall due to the strong inflow of the cross stream towards the symmetric plane. This inward motion carries high momentum fluid from the cross stream to the symmetric plane. Therefore, the axial component velocity profiles gradually smooth out downstream.

5.4 Turbulent Velocity Fields and Characterization

Measurements of turbulent distributions are presented at four downstream locations in the \( x-y \) plane (Fig. 5.19 for \( R = 2 \) and Fig 5.20 for \( R = 4 \)). The turbulent shear stresses \( \overline{uv} \) are compared with calculations using the \( k-\epsilon \) model. The agreement is again less in the initial region. The significant variations of the shear stress distributions clearly show the substantial anisotropy of the flow field. Moving downstream, there is a tendency towards an isotropic flow and the agreement improves.

In Fig 5.19 for \( R = 2 \), the measured shear stresses are considerably larger than the computed results and the profiles exhibit more scatter than those shown in Fig. 5.20 for \( R = 4 \). The relatively higher turbulence level at the near jet exit plane
for $R = 2$, compared to that for $R = 4$, would have contributed to larger and more scattered values of shear stresses (see Fig 5.1). The values of $\bar{u}$ for both velocity ratios considerably exceed those of $\bar{v}$. While the values of $\bar{u}$ and $\bar{v}$ profiles are nearly independent of the velocity ratio, the values of $\bar{uv}$ profiles are strongly dependent. Note the value of the shear stress which is an order of magnitude higher in $R = 4$ than the case of $R = 2$. This increase of the shear stress $\bar{uv}$ with the increasing velocity ratio, which results in increased velocity gradients, reinforces the observation that the mean velocity gradients play important roles in the turbulent energy production.

The $\bar{uv}$ profiles change their sign at the two edges of the jet. One edge exists at the interface sheet between the jet stream and the outer cross stream and the other at the lower edge of the jet cross-section. The locations of the change of sign in $\bar{uv}$ profiles arise generally in the change of sign in the mean velocity gradients (see Fig. 5.17 and Fig. 5.18), especially in $U/U_o$ profiles. Farther downstream, there exists essentially negligible velocity gradient, and thus the $\bar{uv}$ profiles smooth out. This mechanism will be discussed in detail in the next paragraph.

The position of the maximum $\bar{u}$ and $\bar{uv}$ profiles corresponds approximately to the location of the jet cross-section where the velocity gradients, $\frac{\partial u}{\partial x}$ and $\frac{\partial u}{\partial y}$, are maxima for both velocity ratios (see Fig. 5.17 and Fig. 5.18). The $\bar{v}$ profile maximum corresponds to the edges of the jet where $\frac{\partial v}{\partial y}$ is maximum. Recall from the turbulent and mean kinetic energy equation that the maximum transfer of energy from the mean flow to the turbulent flow occurs when $|\rho \bar{u} \bar{u}_j| |\frac{\partial u}{\partial x}|$ is maximum.
Figure 5.19: Normalized Reynolds stress distributions in x-y plane for $R=2$; symbols, measurement, — in (c), prediction
Figure 5.20: Normalized Reynolds stress distributions in x-y plane for R=4; symbols, measurement, — in (c), prediction
Thus, the turbulent stresses which are an indication of the level of turbulence will be a maximum when the Reynolds stress and mean velocity gradient are maxima.

The turbulent kinetic energy dissipation rates calculated from the computed and the experimentally determined velocity fields are presented at $x/D = 4$ and $x/D = 8$ for $R=2$ (Fig. 5.21) and for $R=4$ (Fig. 5.22). The turbulent energy dissipation rate is defined as:

$$\epsilon = \frac{\nu}{2} \left( \frac{\overline{u_i} - \overline{u_j}}{x_j - x_i} \right)^2$$  \hspace{1cm} (5.3)

The numerical results of the dissipation rate $\epsilon$ are obtained by solving the transport equation of $\epsilon$ in the $k-\epsilon$ turbulence model. The dissipation rate expressed in this equation is difficult to measure unless multi-probes are employed. However, an approximated value of the dissipation rate can be obtained experimentally using the isotropic assumption. The dissipation rate for isotropic flow is defined as:

$$\epsilon = 15\nu u^2 / \lambda_T^2$$  \hspace{1cm} (5.4)

where $u^2$ is the longitudinal mean square velocity and $\lambda_T$ is the longitudinal Taylor microscale. The equation implies that the energy dissipation is a small scale phenomenon.

Comparisons of calculated dissipation rate with measured values are presented in Fig. 5.21 and Fig. 5.22. The dissipation rates are normalized with $\rho U_o^3 / D$. In Fig. 5.21 (a) and at $y > 3$, the computed viscous dissipation is greater than the experimentally measured value. This is probably a result of the calculated trajectories of the jet penetrating farther into the confined crossflow than is the actual case.
Figure 5.21: Comparison of dissipation rate between predictions and measurements for $R = 2$; (a) $x/D = 4.0$, (b) $x/D = 8.0$; •, measurement, — , prediction
Figure 5.22: Comparison of dissipation rate between predictions and measurements for $R = 4$; (a) $x/D = 4.0$, (b) $x/D = 8.0$; •, measurement, —, prediction
The fairly poor agreement is due to the nonhomogeneity in the turbulent velocity field. As mentioned in the introduction, turbulence models including the k-ε turbulence cannot yet sufficiently resolve fine scales at practically occurring Reynolds numbers[5]. When modeling the dissipation rate (Eq. 5.3) that appears in the exact (primitive) transport equations into a solvable form, the k-ε model undos influence upon the local mean velocity. This is inappropriate since the dissipation occurs in the finest scales of motion and these fine scales do not reflect the local mean velocity field[63]. The strong dependency of the dissipation rate on the mean flow field is only acceptable at the low Reynolds number flows. Also, the computational results depend on the assumption that $u^2 = v^2 = w^2$. This is not the case in the actual flow. Note further that there is an order of magnitude difference in the viscous dissipation for $R = 4$ as compared to $R = 2$.

Correlation functions are of fundamental importance for the characterization of the turbulent structure. The autocorrelation function determines the length of the past history of velocity fluctuations that is related to a given event. As discussed in Chapter 3, the calculation of the integral length scale and Taylor microscale depends upon the construction of autocorrelation functions of randomly sampled velocity fluctuations in the sparsely seeded flow. This construction has been also extensively discussed in the literature [64-67]. Edwards and Kolodzy[67] developed a method for measuring unbiased autocorrelation functions in sparsely seeded flows. In practice, the autocorrelation function is obtained via a transform method as
Autocorrelation functions of the x-direction velocity fluctuations are constructed at two downstream locations (x/D = 4 and 8) in the symmetric plane in Fig. 5.23 for R = 2 and Fig. 5.24 for R = 4. For R = 2 in Fig. 5.23, the sinusoidal behaviour of the autocorrelation function is evident at y/D = 5 both for x/D = 4 and x/D = 8. A similar observation can be made for R = 4 in Fig. 5.24 but in this case at y/D = 7. This periodic nature represents the passage of the vortex structures downstream. This periodic nature also indicates a preserved eddy structure in this region of flow. The attenuation of the peaks indicates that the strength of the vortical structures decreases downstream. The shapes of the autocorrelation functions with no noticeable peaks again reinforce the turbulent wake behaviors at the points, typically at y/D = 3 and x/D = 8 for R = 2. In a fully turbulent region, the velocity fluctuations are random, and thus the correlation functions do not exhibit the presence of preferred coherent structures.

An important part of the description of the turbulent flow field is the determination of the energy content of wave vectors or frequencies. This can be done by reducing the time-dependent signal into its harmonic components using a spectrum analyzer or other transform methods. The spectrum of instantaneous velocities are obtained by the Fourier transform of the autocorrelation function:

\[ S(f) = \frac{1}{2\pi} \int_{-\infty}^{\infty} R(\tau)exp(-j2\pi f \tau) d\tau \]  \hspace{1cm} (5.5)

The relation is valid only when turbulence in the flow can be considered a station-
Figure 5.23: Autocorrelation functions for R=2 at z/D = 0.0
Figure 5.24: Autocorrelation functions for R=4 at z/D = 0.0
ary process[68]. Here, a stationary process is defined as the case in which the statistical functions describing the random process are time-independent (i.e. the autocorrelation function is dependent only on the delay time). The FFT algorithm originally developed by Cooley and Tukey[54] is used. All spectra are calculated by a fast Fourier transform of autocorrelation functions with a 5.0 kHz cut-off frequency. This cut-off frequency appears as a sharp dip in the spectral density at high frequency values and therefore the cut-off frequency is sufficiently large to cover the frequency range of interest in this study (see Fig. 5.25 for R=2 and Fig. 5.26 for R=4 ). Spectral functions are plotted as a function of relative spectral density in log $S(f)$ versus log $f$ where $f$ is the frequency. Note the frequencies of the peaks in the spectra correspond to the periodicity of the autocorrelation functions, which reveal again the passage of the vortical structures. Also, the decay of the spectra is compared to Kolmogorov's "-5/3 law" in the inertial subrange of the Universal Equilibrium Theory[52].

Turbulent scales are of fundamental importance in the characterization of turbulent flows and also in the formulation of turbulence model. Integral time scales, $T$, are plotted versus the vertical direction, $y/D$, in the x-y plane of symmetry in Fig. 5.27. The integral time scale is an experimental estimate of the largest time scale of the turbulent flow field. The magnitude of the time scale increases downward and also increases downstream at a reduced rate. Also, a higher velocity ratio, $R$, generally results in a higher value of the time scale. Physically, this indicates
Figure 5.25: Auto-spectral densities for \( R=2 \) at \( z/\Gamma = 0.0 \)
Figure 5.26: Auto-spectral densities for R=4 at z/D = 0.0
that the wake caused by the jet extends farther downstream.

Integral length scales, $L$, are graphed versus vertical displacement $y/D$ in the $x$-$y$ plane of symmetry in Fig. 5.28. The integral length scale represents the largest turbulent scale in the flow field. As introduced in Chapter 3 (Eq. 3.15), the integral length scale is obtained by integrating the spatial correlation function up to the first zero crossing of the spatial axis. When only a one-point measurement system is applicable, the spatial correlation function is usually calculated by use of Taylor's hypothesis. The basic tenet of the hypothesis is that a time varying function of any statistical property can be converted to the spatially varying function when the flow is stationary and the turbulence intensity is not large. Thus, the credibility of the Taylor hypothesis is dependent on the local flow situation.

In an attempt to improve the physical understanding of the results, the bar graphs are also presented in Fig. 5.28. The flow field in the $x$-$y$ plane of symmetry is divided into four distinct region: (1) the wake region, located near the lower wall, and on the leeside of the jet, (2) the entrainment region, where mixing between the jet and the free stream is dominant, (3) the central region, serving as a transition region between the wake and entrainment region, and (4) the free stream region, located outside the jet's influence. The length scale generally increases as $x/D$ and $R$ increases, and decreases as $y/D$ increases. The length scale is considerably higher in the wake region and in the central region than the free stream region. The variations of length scales indicate that the flow field is anisotropic and non-homogeneous.
Figure 5.27: Integral time scales at $z/D = 0.0$ ; (a) $R=2$, (b) $R=4$
Figure 5.28: Integral length scales at $z/D = 0.0$; (a) $R=2$, (b) $R=4$
Figure 5.29: Taylor microscales at $z/D = 0.0$; (a) $R=2$, (b) $R=4$
Profiles of the Taylor microscale, $\lambda_T$, are plotted in Fig. 5.29 as functions of the vertical position $y/D$ for two downstream locations for $R = 2$ and 4. Physically, the Taylor microscale is a measure of the average dimension of the eddies which are primarily responsible for energy dissipation. The terminology of the dissipation scale or the Taylor microscale aminates from this physical reason. For isotropic turbulence\cite{46}:

$$\frac{1}{\lambda_T^2} = \frac{1/5 \int_0^\infty k^4 E(k)dk}{\int_0^\infty E(k)dk}$$

where $k$ is a wave vector of the velocity fluctuation and $E(k)$ is the energy spectrum function. This expression may be interpreted physically as the ratio of the energy dissipation rate of the turbulence to the total turbulent kinetic energy. As $\lambda_T$ increases, the dissipation rate therefore decreases. Consider the results shown in Fig. 5.29. The value of $\lambda_T$ increases downward. But, the increasing rate is is slower than the rate of the integral scale shown in Fig. 5.28. This implies the relatively higher rate of the energy production downward. The $\lambda_T$ is much more strongly influenced by the downstream location, $x/D$, for the larger velocity ratio, $R=4$. The microscale consistantly increases with increasing values of $x/D$ for $R=4$. The microscale of this flow generally ranges from 5 to 40% of the integral scale.

The motivation for determining probability density functions (PDF) in this study arises from the non-homogeneous nature of the flow. Before presenting results, the basic relationship between the probability function and the nature of the non-homogeneous turbulent flow field will be discussed.
According to Townsend[3], the probability density function is nearly normal in homogeneous turbulence, less normal near the center of a shear flow, and very far from normal near the edge of a free turbulent flow such as a jet. The departures are of two kinds: (1) skewness of the distribution with non-zero odd movements, and (2) distortion leading to abnormally large values for higher order even movements. The skewness of the distributions is connected to the convection of turbulent energy by turbulent movements. An abnormally large flatness factor indicates that the distribution of the intensity of the quantity is spotty. The probability function also effectively illustrates the nature of intermittent turbulent flows or mixing cases between two or more different types of fluids. The turbulent flow field under study contains these two types of flow characteristics.

The probability density function of \( u(t) \) occurring in the range of \( u_1 \leq u(t) \leq u_2 \) is defined by

\[
P(u) = \lim_{\Delta u \to 0} \frac{\text{Prob}[u_1 \leq u(t) \leq u_2]}{\Delta u}
\]  

(5.7)

where \( \text{Prob}[u_1 \leq u(t) \leq u_2] \) is the ratio of total time of \( u(t) \) lying within the window \( \Delta u(=u_2-u_1) \) to the total observation time. Experimentally, the probability density functions are calculated from 2560 velocity observations at each location using 16 bins equally spaced over the 4 \( \sigma \) limits of the data (\( \Delta u = 0.5\sigma \)).

Fig 5.30 presents the typical probability density functions of the longitudinal component of the instantaneous velocities measured at \( x/D = 4 \) in the plane of symmetry for \( R=2 \). In Fig. 5.31, similar plots for \( R=4 \) at \( y/D = 3, 5, \text{and } 7 \) are
presented. The instantaneous velocity $\bar{U}$ is nondimensionalized with the mean value and standard deviation. At $y/D = 2$ for $R = 2$, the negatively skewed PDF (skewed to the lower velocity direction) indicates that the contribution of the slowly moving fluid elements of the wake flow is higher than the contribution of the relatively high speed fluid elements of the cross flow. The PDF profile at $y/D = 5$ for $R = 4$ (Fig. 5.31) is slightly skewed negatively, indicating that the wake region further extends upward with the increasing velocity ratio, compared to the profile for $R = 2$ at $y/D = 4$.

The measured skewness profiles at $x/D = 4$ for $R = 2$ and 4 are shown in Fig. 5.32. Recall that the skewness $S_u$ is zero in the isotropic flows. For $R = 2$, $S_u$ is slightly positive near the bottom wall, changes sign to become negative and then approaches zero as $y/D$ increases. For $R = 4$, $S_u$ is larger and positive initially and remains positive farther into the flow field before eventually becoming negative. Skewed distributions are typical of flows with strong gradients of turbulence intensity. A comparison between the skewness variation and turbulence intensities (Fig. 5.20 (a)) at $x/D = 4$ for $R = 4$ provides insight into the nature of the flow. In fact, the shapes of the curves are identical. Increasing skewness occurs in regions of increasing turbulent intensity. Conversely, decreasing skewness parallels flow regions of decreasing turbulent intensity. Outside of the influence of the wake region, the maxima of skewness and turbulent intensity both occur approximately at $y/D = 5.0$. 
Figure 5.30: Measured probability distributions of x-component velocity at $x/D=4$ and $z/D=0.0$ for $Re=2$; (a) $y/D=2$, (b) $y/D=4$, (c) $y/D=6$
Figure 5.31: Measured probability distributions of x-component velocity at $x/D = 4$ and $z/D = 0.0$ for $R=4$; (a) $y/D=3$, (b) $y/D=5$, (c) $y/D=7$
Figure 5.32: Measured skewness factor profiles of x-component velocity at $x/D=4.0$ and $z/D=0.0$ for $R=2$ and 4
Figure 5.33: Measured flatness factor profiles of x-component velocity at x/D = 4.0 and z/D = 0.0 for R = 2 and 4
One effect of the inhomogeneous nature of the turbulent intensity is that the flatness factor exceeds its normal distribution value of 3.0. Consider the results depicted in Fig. 5.33. For R=4, the flatness factor \( F_u \) becomes exceptionally large in the central region of the flow. The location of \( F_{um} \) occurs once again at the location of \( S_{um} \) and \( (\bar{\omega}^{1/2} / U_o)_{max} \) (see Fig. 5.20 (a) for turbulence value at \( x/D = 4.0 \)). Large flatness factors usually indicate that the distribution of the turbulent quantities is spotty. The small scale components of the velocity field are noticeably spotty or intermittent, and the effect is more pronounced the greater the difference in sizes between the energy containing and dissipation regions of the spectrum\[3\].

Kolmogorov\[69\] stated that the transfer of energy from large to small eddies by a cascade process means that spatial fluctuations in the rate of energy transfer from eddies of a particular size bias the rate of transfer at the next step in the cascade. The result is that some parts of the flow are regions of large dissipation while others have a very low rate of dissipation. The consequence is the spottiness indicated by large flatness factors values. The flow of a turbulent jet in a confined crossflow demonstrates the existence of the spottiness described by Townsend and Kolmogorov for the region of highest turbulence intensity.
Chapter 6

Summary

An experimental and computational investigation has been made for the turbulent flow of the jet in a confined cross flow. The significant results will now be summarized.

Detailed mappings of the mean velocities, species concentration and mean vorticities were made for jet to cross flow velocity ratios (R) equal to 2, 4 and 6. The velocity ratio value has a strong influence on the development of the flow field downstream and the existence of the jet impingement. For the lower velocity ratio (R = 2), the jet is bent down rapidly by the cross stream. In this case, no impingement occurs on the top wall for the separation of the confining surfaces equal to ten jet diameters. For the larger velocity ratios (R = 4 and 6), significant upward motion continues downstream, and the jet directly impinges on the top wall, and after impingement a three-dimensional wall jet-type flow develops along the top wall.

A wake region exists immediately downstream of the jet discharge and the wake region extends further downstream with increasing velocity ratio. Far downstream,
the fluids in the wake region gradually gain the axial momentum as the cross stream carries a higher momentum from the outer region to the plane of symmetry. A counter-rotating vortex pair is predicted. The vortex structure continues to exist after impingement while gradually losing its strength downstream.

Turbulent intensities and shear stresses were measured at several downstream locations in the plane of symmetry. The turbulent flow field is highly anisotropic in the initial region. There is a tendency towards isotropy farther downstream. The jet's initial condition has a direct influence on the turbulent flow field downstream. While the turbulence intensities are not strongly dependent on the velocity ratio, the turbulent shear stresses for $R=4$ are an order of magnitude higher than those for $R=2$.

The structured nature of the turbulent flow field has been documented from statistical measurements at two downstream locations $(x/D = 4$ and 8) in the plane of symmetry. The passage of vortical, coherent structures is observed from the characteristic behaviors of the autocorrelation functions. Significant variations of the integral and Taylor microscales have revealed the strong, anisotropic, and non-homogeneous nature of the turbulent flow field. The turbulent flow field can be characterized by large-scale phenomena. The energy generation rate is considerably higher than the dissipation rate of the kinetic energy. Probability density functions, skewness and flatness factors point out the large intermittent and spotty nature of the flow field.
Calculation results were compared to the experimentally determined values. The two-equation model generally overestimates the effective value of R. The jet in the computational model penetrates further into the outer stream than is the actual case. Agreement of mean velocity and turbulent shear stress comparison is only fair in the initial region. The agreement improves as the flow field tends towards isotropy farther downstream. The relatively coarse grids used, which cannot sufficiently resolve the significant variations of variables in all three-dimensional coordinates, are primarily responsible for the difference. Considerable discrepancies between the measured and computed dissipation rates are observed. The primary reason arises in the relatively low resolution of the two-equation turbulence model in predicting the fine scale turbulence structure.
Bibliography


Vita

Keun S. Chang was born in October 20, 1956 in Kunggi Providence in Korea. At age of fifteen, he left his home town for his high school education from Seoul High School in Seoul, Korea, and graduated in February, 1975. He then received his Bachelor Degree of Mechanical Engineering from Seoul National University in Seoul, in February, 1979. After his college education, he worked as a research engineer in Agency for the Defense Development of Korea for five years. In August, 1984, he enrolled in Mechanical Engineering at Louisiana State University for his graduate study. He received the degree of Master of Science in Mechanical Engineering, in May, 1986. He is now a candidate for the degree of Doctor of Philosophy in Mechanical Engineering to be awarded at the commencement in August, 1989.
DOCTORAL EXAMINATION AND DISSERTATION REPORT

Candidate: Keun Sun Chang

Major Field: Mechanical Engineering

Title of Dissertation: An Experimental and Computational Investigation of Turbulent Jet Impingement in a Confined Cross Flow

Approved:

[Signatures]

Dean of the Graduate School

EXAMINING COMMITTEE:

[Signatures]

Date of Examination:

May 12, 1989