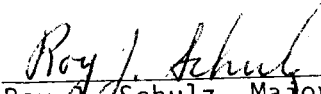
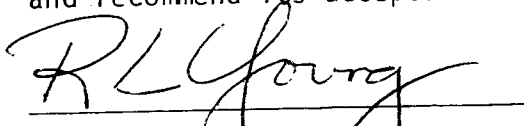
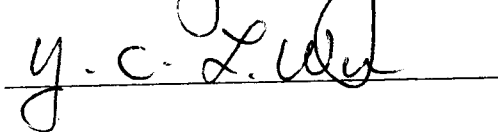


To the Graduate Council:

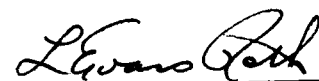
I am submitting herewith a thesis written by N. S. Vidyasagaran entitled "Study of Boundary Layers Approaching Separation in Diffusers." I have examined the final copy of this thesis for form and content and recommend that it be accepted in partial fulfillment of the requirements for the degree of Master of Science, with a major in Mechanical Engineering.


Roy J. Schulz, Major Professor

We have read this thesis
and recommend its acceptance:

Accepted for the Council:


Vice Chancellor
Graduate Studies and Research

STUDY OF BOUNDARY LAYERS APPROACHING
SEPARATION IN DIFFUSERS

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

N. S. Vidyasagaran

December 1981

3055570

ACKNOWLEDGEMENTS

The valuable guidance, suggestions, and encouragement given by Dr. Roy J. Schulz, Major Professor are acknowledged with deep gratitude. The encouragement and support given by Dr. Ying-Chu Lin Wu during the course of this investigation is highly appreciated. The author is very grateful to Dr. R.L. Young for his tutorial guidance and encouragement during the past two years of graduate work. The technical assistance given by Dr. Frank Collins is acknowledged.

The author is grateful to Dr. G.H. Masapati for his help in analyzing the computer program and the help given by Mr. P. Sundaram is very much appreciated. The author acknowledges the timely assistance of the UTSI Computer Center. Appreciation is also expressed to Mrs. Debbie LeMaster who typed the draft and to Mrs. Charlotte Henley and Mrs. Genna Luke who typed the final version of this thesis.

ABSTRACT

The purpose of this study was to develop and evaluate a numerical technique for predicting the performance of unstalled diffusers for a magnetohydrodynamic power generating cycle being investigated at University of Tennessee Space Institute. The work involved two tasks. The first task was to conduct a survey of literature to find and catalogue families profiles of velocity and temperature for flow near to smooth and rough walls. The second task was to use the appropriate law-of-the-wall to serve as boundary conditions in a diffuser flow computer program developed by Harsha and Glassman (1). The Harsha-Glassman program is based on a one-equation, turbulent kinetic energy model for turbulence transport that uses a prescribed length-scale for kinetic energy dissipation. In this study, the dissipation length scale was modified to account for adverse pressure gradient effects. The modified version of the computer code predicts velocity profile in the boundary layer very close to the standard data. The modified version of the code predicts separation closer to reality than the original version.

TABLE OF CONTENTS

CHAPTER	PAGE
I. INTRODUCTION	1
Purpose of the Research	1
Discussion of Diffusers for MHD Application	3
Supersonic Diffuser	4
Subsonic Diffuser	5
Summary of Discussion	9
Diffuser Performance Parameters	9
Diffuser Pressure Recovery	11
Blockage Parameter B	11
Geometric Parameters	12
Additional Inlet Parameters	12
Overall Diffuser Effectiveness	13
Pressure Recovery Factor	13
Viscosity and Temperature Effects on Real Diffuser Performance	13
II. LITERATURE SURVEY	15
Subsonic Diffuser Performance and Design Criteria	15
Aspect Ratio Effects	18
Cross-Sectional Shape Effects	18
Reynold's Number Effects	19
Inlet Mach Number Effects	20
Inlet Turbulence	20
Inlet Velocity Profile Distortion	20

CHAPTER	PAGE
Tail Pipes	21
Wall Fairing	21
Velocity Distribution in MHD Diffusers	23
Boundary Layer Velocity Profiles	24
Velocity Profile on Rough Walls	31
Thermal Boundary Layers	34
Reynold's Analogy.	35
Thermal Boundary Layer Profile Functions	37
Turbulence Transport Model	40
III. THEORY AND NUMERICAL SOLUTION PROCEDURE.	44
Governing Equations	44
Initial Profiles and Boundary Conditions	48
Boundary Conditions.	48
Auxiliary Relationships	49
Theory of Law-of-the-Wall	51
Treatment of the Wall Region	58
Modifications on TKE Dissipation Length Scale	
Distribution	62
Mixing Length Distribution in the Wall Region	64
Numerical Solution Procedure	66
Coordinate System	67

CHAPTER	PAGE
The Finite Difference Equations	67
Treatment of the Edge Region	74
Solution of the Finite Difference Equations	76
Calculation of Cross Stream Distance	77
Treatment of Rectangular Geometry	78
IV. RESULTS AND DISCUSSION	81
Results	82
Performance Prediction for Model Diffuser	101
V. CONCLUSIONS AND RECOMMENDATIONS	108
Conclusions	108
Recommendations for Further Work	109
LIST OF REFERENCES	110
APPENDIX A	114
VITA	116

LIST OF FIGURES

FIGURE	PAGE
1. Schematic of Diffuser Flow	6
2. Geometry for Two Dimensional Straight Wall Diffuser. . . .	8
3. Diffuser Flow Regimes as Established by Fox and Kline (1962) for Two Dimensional Diffusers	10
4. Wall Fairing	22
5. Possible Modes of Log-Law Breakdown in Adverse Pressure Gradients	29
6. Temperature Profile for Flow of Air ($Pr=0.7$) Along a Plate With Constant Free Stream Velocity and Constant Surface Temperature	38
7. Length Scale Distribution in Diffuser Boundary Layer Assumed by Harsha and Glassman(1)	43
8. Coordinate System	45
9. Viscosity Variation in the Laminar Sublayer for Positive dp/dx	56
10. Viscosity Variation in the Log Layer for Positive dp/dx . .	57
11. Schematic of Subscripts for the True and Slip Values of	59
12. Illustration of the Unsuitability of a Cartesian Grid for a Growing Boundary Layer	68
13. A Small Part of Grid Showing a Typical Control Volume . .	70

FIGURE	PAGE
14. Illustration of Control Volume Considered in the Deriation of Φ_2	75
15. Schematic of Area Transformation	79
16. Comparison of Power Law Profile with Actual Profile	84
17. Comparison of Logarithmic Velocity Distribution at x = 8.299 Inches	85
18. Comparison of Logarithmic Velocity Distribution at x = 9.299 Inches	86
19. Comparison of Logarithmic Velocity Distribution at x = 11.799 Inches	87
20. Comparison of Logarithmic Velocity Distribution at x = 15.049 Inches	88
21. Comparison of Logarithmic Velocity Distribution at x = 17.249 Inches	89
22. Comparison of Logarithmic Velocity Distribution at x = 18.449 Inches	90
23. Comparison of Logarithmic Velocity Distribution at x = 22.749 Inches	91
24. Comparison of Logarithmic Velocity Distribution at x = 24.149 Inches	92
25. Comparison of Theoretical and Experimental Velocity Distribution at x = 8.29 Inches	93

FIGURE	PAGE
26. Comparison of Theoretical and Experimental Velocity Distribution at $x = 9.29$ Inches	94
27. Comparison of Theoretical and Experimental Velocity Distribution at $x = 11.79$ Inches	95
28. Comparison of Theoretical and Experimental Velocity Distribution at $x = 15.04$ Inches	96
29. Comparison of Theoretical and Experimental Velocity Distribution at $x = 17.24$ Inches	97
30. Comparison of Theoretical and Experimental Velocity Distribution at $x = 18.449$ Inches	98
31. Comparison of Theoretical and Experimental Velocity Distribution at $x = 20.749$ Inches	99
32. Comparison of Theoretical and Experimental Velocity Distribution at $x = 24.149$ Inches	100
33. Comparison of Theoretical and Experimental Values of C_f Along Diffuser Length	102
34. Comparison of Theoretical and Experimental Values of Shape Factor (H) Along Diffuser Length	103
35. Comparison of Theoretical and Experimental Values of Momentum Thickness (θ) Along Diffuser Length	104
36. Predicted Diffuser Performance Map for Model Diffuser Using Modified Code	105
37. Predicted Diffuser Performance Map for Model Diffuser Using Original Code	106

LIST OF SYMBOLS

$A_{\text{effective}}$	Effective flow area
$A_{\text{geometrical}}$	Geometrical flow area
a_1	Constant in turbulent shear stress model
a_2	Turbulent kinetic energy dissipation constant
A_t	Damping constant in van Driests formula
AS	Aspect ratio
AR	Area ratio
B	Blockage parameter
b	Inlet or exit heights
C_f	Drag coefficient
C_p	Pressure recovery coefficient, or specific heat at constant pressure
C_{pi}	Ideal one dimensional pressure recovery coefficient
D	Dissipation integral
d	Source term
d	Effective height of roughness element
ds	Equivalent sand grain roughness
E	Integration constant, Roughness length scale
F	Pressure gradient parameter
H	Shape factor
h	Enthalpy
$\tilde{h}$	Total enthalpy
J_j	Diffusional flux of chemical species j

k	Von Karman constant
K	Turbulent kinetic energy (TKE)
l	Mixing length
l_p	Prandtl mixing length
l_k	Length scale for TKE dissipation
$\dot{m}''$	Mass flow across any layer
m_j	Mass fraction of chemical species j
M	Mach number
N	Number of strips across layer
p	Static pressure
p_e	Exit pressure
Pr	Prandtl number
Pr_{eff}	Effective Prandtl number
Pr_k	Turbulent Prandtl number for turbulent kinetic energy
Pr_t	Turbulent Prandtl number
P_t	Centerline static pressure
P_{ot}	Centerline stagnation pressure
q	Dynamic pressure ($1/2\rho u^2$)
q_w	Heat flux at the wall
r	Distance from axis of symmetry
Re	Reynolds number
Re_θ	Reynolds number based on momentum thickness
s	Dimensionless shear stress at the wall
St	Stanton number

t	Absolute temperature
t_w	Temperature of fluid at the wall
u	Velocity in the longitudinal direction
u^+	Dimensionless velocity
u_τ	Friction velocity
u'	Fluctuating component of u velocity
U	Mass average or bulk flow velocity
v	Velocity in y direction
v_r	Velocity in radial direction in axisymmetric co-ordinates
v'	Fluctuating component of v velocity
W	Width
w	Velocity normal to u and v
w'	Fluctuating component of w velocity
x	Coordinate along diffuser axis, Distance in streamwise direction
y	Distance in cross-stream direction
y^+	Non-dimensional y
z	Co-ordinate normal to x and y

GREEK SYMBOLS

α	Area ratio A_2/A_1
γ	Ratio of specific heats
δ	Boundary layer thickness
δ_1	Thickness of sub-layer
δ^*	Displacement thickness

Δ	Defect thickness
ΔB	Profile shift parameter
μ	Molecular viscosity
μ_t	Turbulent viscosity
μ_{eff}	Effective viscosity
θ	Divergence angle, Momentum thickness
ξ	Overall diffuser effectiveness
ρ	Density
τ_t	Turbulent shear stress
τ_w	Shear stress at the wall
ϕ	A general dependent variable
ψ	Stream function
ω	Dimensionless stream function

Subscripts

1	Entrance/throat
2	Discharge
ad	Adiabatic
e	Edge of the boundary layer
E	External boundary of the layer
h	Pertaining to enthalpy
I	Internal boundary of the layer
j	Pertaining to chemical species j
S	Wall boundary
t	Fully turbulent

∞ Freestream value

Superscripts

- Time averaged quantity

' Fluctuating component

Additional subscripts and superscripts defined in text.

CHAPTER I

INTRODUCTION

Purpose of the Research

The purpose of this research was to develop and evaluate a numerical technique for predicting the performance of unstalled diffusers for a Magnetohydrodynamic (MHD) power generating cycle being investigated at The University of Tennessee Space Institute (UTSI). The research is being done by the Energy Conversion Division (ECD) of UTSI for the United States Department of Energy under contract No. DE-AC02-79-ET10815. The power cycle being investigated and developed at UTSI is based on burning seeded (K_2CO_3) pulverized coal to very high temperatures ($3000^\circ K$) and expanding the plasma through an MHD generator channel to extract electrical power directly. However, the gas leaving the generator is still hot ($2500^\circ K$) and high speed with low static pressure (subatmospheric). Therefore, a diffuser is a critical flow train component that interfaces the channel with the flow train components downstream which operate at atmospheric pressure and serve to remove heat, slag, and exhaust gas pollutants from the flow. Therefore, it is necessary to develop a technology to permit design assessment, evaluation and performance extrapolation for diffusers for the MHD application.

To develop this technology, two tasks were identified. The first first task was to conduct a survey of literature to find and catalogue families of profiles of velocity and temperature for flow

near to smooth and rough surfaces. These profiles are typically of the form where u^+ and t^+ are scaled velocity and temperature, respectively:

$$u^+ = \frac{1}{k} \log_e \left(\frac{u_\tau y}{\nu} \right) + B \left(\frac{u_\tau d}{\nu} \right) \quad (1.1)$$

$$t^+ = \frac{1}{k(Pr_t)} \log_e \left(\frac{u_\tau y}{\nu} \right) + B \left(\frac{u_\tau d}{\nu}, Pr_t \right) \quad (1.2)$$

These profiles describe the behavior of flow from the viscous sublayer to the fully turbulent main external flow field. Besides providing direct information about the shapes of velocity and temperature profiles in these regimes, they also serve as important 'matching' functions when numerically modeling flows. For example, they serve to match the outer or main external flow conditions to the conditions essentially at the wall. This means that the numerical solution procedure is freed from the necessity of providing a very fine mesh near the wall to resolve the physics of these high gradient regimes. Thus, a great economy is achieved in numerical solution techniques, when law-of-the-wall matching procedures are used for boundary conditions. These profiles also serve to describe the momentum and heat flux to the walls.

The second task of the research program was to use the appropriate law-of-the-wall families to serve as boundary conditions in a diffuser flow computer program.

The computer program with the appropriate law-of-the-wall matching procedure was utilized to evaluate the performance of a diffuser for the Coal-Fired Flow Facility, which was experimentally investigated in a model study using isothermal flows. Conclusions were drawn on the validity of this technique to predict unstalled diffuser performance.

Discussion of Diffusers for MHD Application

The primary function of a diffuser is to recover the inherent pressure potential of the flow as it enters the diffuser. An ideal diffuser will recover all of this pressure potential; however, real diffusers only recover a portion. Diffusers are key elements in power systems using expansion devices such as magnetohydrodynamic generator channels. By attaching a diffuser to the end of an MHD channel, the pressure at the exit of the channel can be reduced below atmospheric pressure and the diffuser used to bring the pressure of the flow to ambient conditions. Since the power output of the channel is a function of the pressure ratio across it, an MHD diffuser increases the performance of the channel while simultaneously reducing the flow speed to prevent damage to the components downstream of the diffuser.

The performance of the diffuser is determined by constraints imposed by the operating environment in the open cycle MHD system. They are:

1. High boundary layer blockage at the diffuser inlet.
2. Unsymmetrical velocity distribution due to the magnetic field acting on the plasma.

3. The presence of condensed phase slag particles and very high temperatures which restrict the use of diffuser performance improvement devices such as ring aerofoils, etc, which are typically immersed in the flow.

4. Deposition of slag on the diffuser surface, which makes the surface irregular and wavy and hence difficult to formulate the wall boundary conditions when numerically modeling the diffuser flow.

5. Effect of the diffuser performance and design on the radiant boiler and slag recovery section of the steam generator into which the diffuser usually exhausts in most MHD concepts.

The MHD generator may be operated with supersonic or subsonic flow and both conditions require diffusers for pressure recovery. Since the presence of condensed phase complicates the use of performance improvement devices, and the temperature of the gas is so high, a simple straight wall diffuser system is used in practice. The cost of manufacture and heat and frictional losses are proportional to the length of the diffuser. So, the shortest diffuser system with minimum heat and frictional losses giving the maximum static pressure recovery is desired for a direct, coal-fired MHD system such as that being investigated at the University of Tennessee Space Institute.

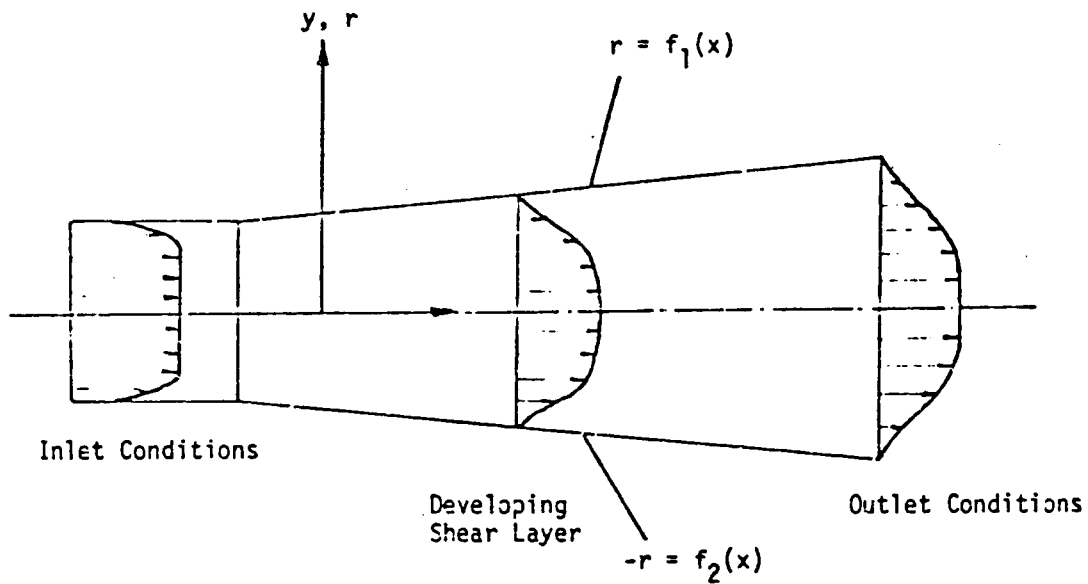
Supersonic Diffuser

The two main types of supersonic diffusers with fixed geometry are (a) the convergent-divergent diffuser and (b) the constant area

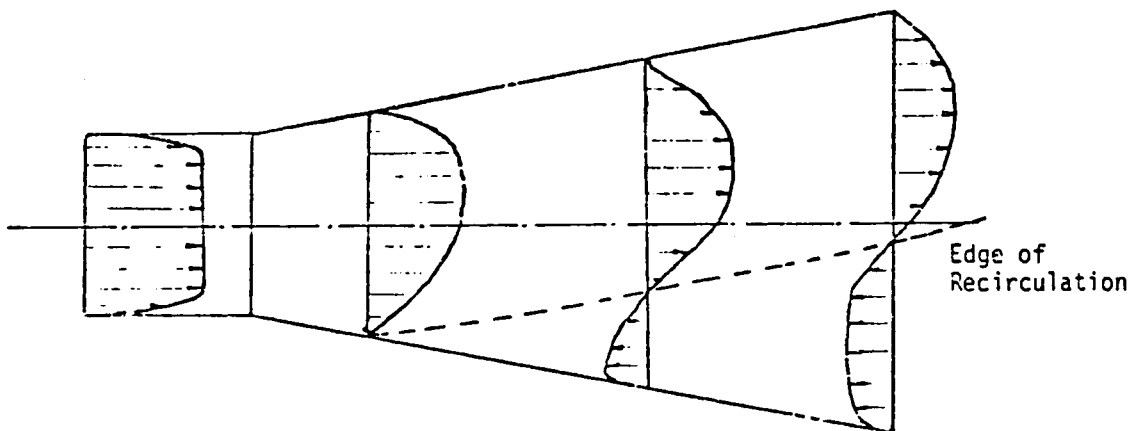
diffuser. The convergent-divergent diffuser is very sensitive to the inlet conditions and to off-design generator operating conditions. However, this configuration is very efficient when the shock occurs at the diffuser throat (at the lowest Mach number). Since this can be achieved only at one particular operating condition, and since MHD generators have to operate over a wide range of flow conditions, the convergent-divergent diffuser is not suitable for MHD application. Though the constant area diffuser is geometrically simple, the flow field is very complex due to the shock system through which the transition from supersonic flow to subsonic flow takes place. This process is, in turn, highly dependent upon the boundary layer entering and developing in the diffuser. The transition may occur through a single normal shock, but usually occurs through a series of oblique shocks interacting with the boundary layer. Because the shock system interacts with the relatively thick boundary layer, it is difficult to analytically describe the flow pattern by means of a simple mathematical model.

Subsonic Diffuser

Subsonic diffusers are generally ducts of expanding cross-sectional area. Ideally, the area ratio of the diffuser geometry is the only parameter of importance. However, viscous effects determine the real diffuser performance which is always less than the ideal and makes the problem dependent on the fluid mechanics of the flow in the geometry connecting the inlet and exit planes. Figure 1a gives a



1a. Schematic of Unseparated Flow in an Arbitrary Non-Symmetric Diffuser



1b. Schematic of Separated Flow in an Arbitrary Non-Symmetric Diffuser

Figure 1. Schematic of Diffuser Flow

schematic of an unseparated flow in a subsonic diffuser and Figure 1b gives a schematic of a "stalled" diffuser flow. The geometrical notation is given in Figure 2.

Two main features of interest in a diffuser flow are (a) internal flow pattern and (b) pressure recovery performance. The flow regimes in a diffuser can be classified as follows:

1. No Appreciable Stall

This flow regime is confined to small expansion angles and area ratios. The flow is steady, and although the boundary layers may become thick, at most only small areas of stall or flow separation are observed. The pressure and velocity profiles are essentially symmetrical about the symmetry planes.

2. Large Transitory Stall

In this flow regime the flow is very erratic, and gross fluctuations of the whole flow pattern are observed. The flow corresponds to randomly separating and reattaching flow, and in circular cross-sectional diffusers, the separations and reattachment can exist in a rotating or swirling mode. This flow behavior causes relatively large pressure fluctuations throughout the diffuser.

3. Two Dimensional Stall

In two dimensional stall, the flow separates near the throat and follows one wall; a steady fixed recirculation or stall zone covers the other diverging wall. The flow is steady except for the intense turbulence in the mixing zone between the attached and separated flow.

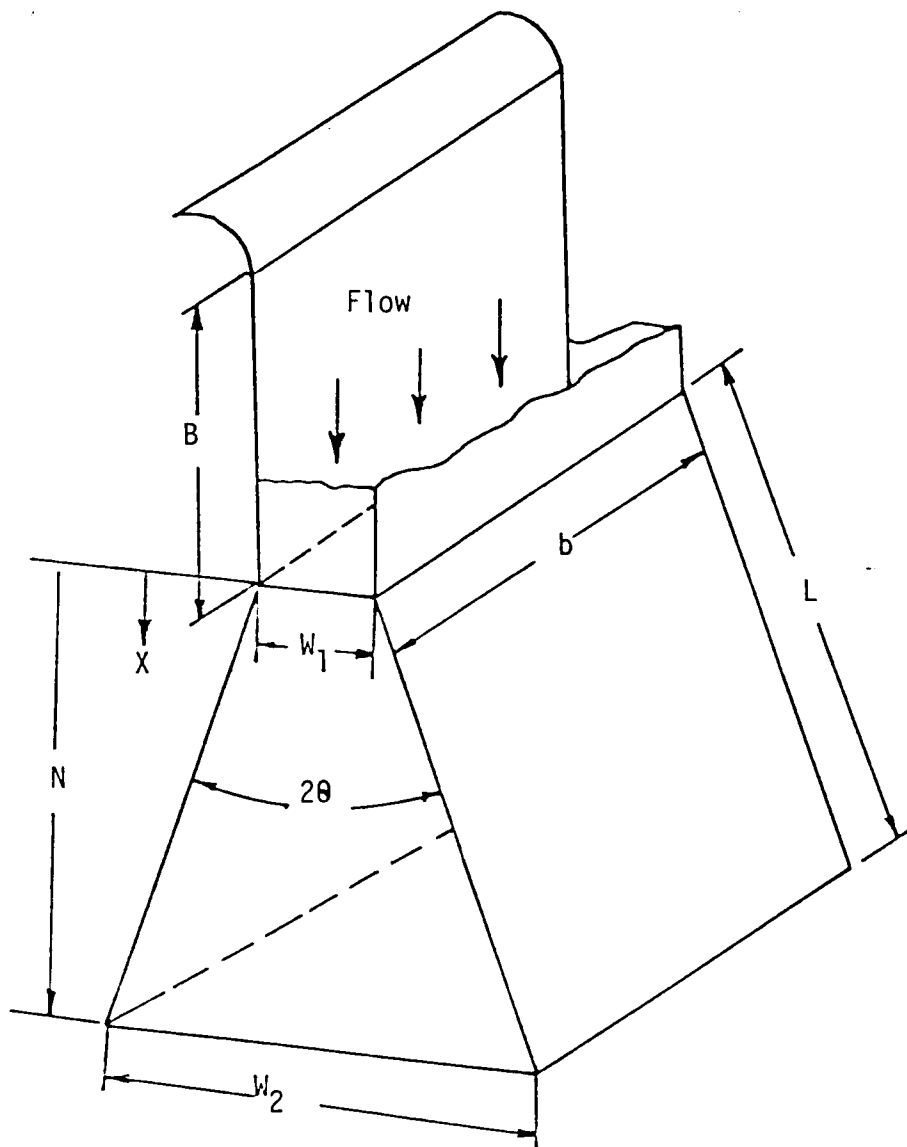


Figure 2. Geometry for Two Dimensional Straight Wall Diffuser

The stall is bi-stable; that is, the state is stable on either diverging wall. The stall can be switched from one diverging wall to the other by an appreciable disturbance at the inlet or inside of the diffuser.

4. Jet Flow

In the jet flow pattern, the incoming stream separates from both diverging walls at or very near the throat and proceeds straight down the diffuser; a large fixed stall covers each diverging wall. The various flow regimes in a diffuser are given in Figure 3.

Summary of Discussion

Even though the diffuser appears to be mechanically or geometrically simple, it is not yet sufficiently understood from a fluid mechanics standpoint. There are several fundamental reasons for this discrepancy. Principal among these is the fact that practical diffusers have various aspect ratios, are three dimensional in shape, and can contain either boundary layer or separated flow over a significant part of the cross-sectional area. Viscous behavior of the fluid combined with the rectangular cross-sectional geometry of most MHD diffusers leads to three dimensional corner effects and unsteady flow which greatly complicates our analytical understanding of the fluid mechanical behavior.

Diffuser Performance Parameters

In the following section a brief description is given of the performance parameters of a subsonic diffuser.

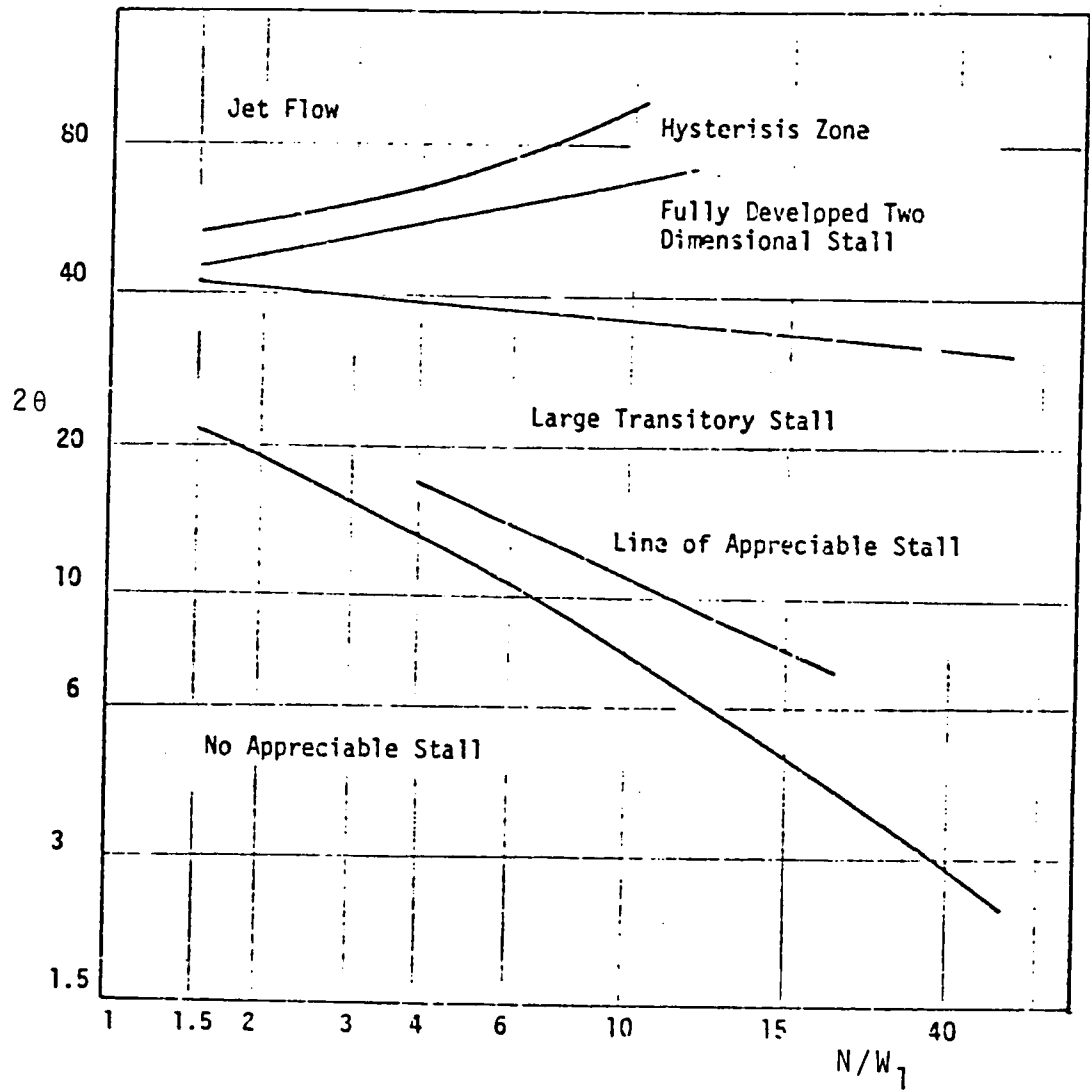


Figure 3. Diffuser Flow Regimes as Established by Fox and Kline (1962) for Two Dimensional Diffusers

Diffuser Pressure Recovery

Diffuser static pressure recovery coefficient, C_p , is defined as the ratio of the pressure rise between the inlet and exit of a diffusing passage to the rise that would be obtained with one dimensional, inviscid, isentropic flow in a diffuser of infinite area ratio having the same inlet centerline velocity.

$$C_p = \frac{p_e - p_t}{p_{ot} - p_t} = \frac{p_e/p_{ot} - p_t/p_{ot}}{1 - p_t/p_{ot}} = \frac{p_e/p_{ot} - p_t/p_{ot}}{1 - \frac{1}{\left(1 + \frac{\gamma-1}{2} M^2\right)^{\frac{\gamma}{\gamma-1}}} \quad (1.3)$$

For one dimensional isentropic flow the amount of diffusion between inlet and exit is dependent only upon the area change and the inlet Mach number and so diffuser length and cross-sectional area are not factors in ideal diffuser performance. Since the area ratio is independent of the exact geometry of the diffuser (independent of whether the diffuser is straight walled, singly or doubly diverging walled, conical in cross-section, etc.), the ideal one dimensional pressure rise is the maximum possible recovery for all diffuser shapes.

Blockage Parameter B

Many applications of diffusers occur where the inlet flow consists of a potential (irrotational) core flow surrounded by viscous boundary layers. An equivalent one dimensional "flow area" can be

calculated based on the core centerline conditions. This effective area is that required to pass a one dimensional flow equal to the actual mass flow with the measured values of throat centerline stagnation pressure and temperature. The blockage is defined as the difference between the actual geometric throat area to the equivalent one dimensional flow area. The blockage parameter B is defined as

$$B = 1 - \frac{A_{\text{effective}}}{A_{\text{geometrical}}} \quad (1.4)$$

Geometric Parameters

The geometry parameters that describe a two dimensional diffuser were given in Figure 2. Three parameters characterize real diffuser geometry for straight wall, single divergence diffusers.

Aspect ratio AS = b/W_1

Area ratio AR = $bW_2/bW_1 = W_2/W_1$

Double divergence angle = 2θ

Additional Inlet Parameters

Depending upon the diffuser application, additional inlet flow parameters are often important for correlating diffuser pressure recovery performance. These parameters are:

1. Inlet Reynold's number (Re_D) based on centerline mean velocity, fluid kinematic viscosity and throat hydraulic diameter.
2. Turbulence level of the inlet flow.

3. Nearness of inlet velocity profile to separation (usually characterized by velocity profile shape factor H).
4. Non-uniform velocity profile in the central core flow entering the diffuser.
5. Distribution of displacement thickness δ^* around the diffuser inlet periphery.
6. Transient characteristics of the inlet flow.

Overall Diffuser Effectiveness (ONE)

The overall diffuser effectiveness is another parameter generally used to describe diffuser performance. This is the ratio of the pressure rise in a real diffuser to that achievable from a diffuser of equal area ratio passing one dimensional, incompressible, inviscid flow at the same flow rate.

$$\xi = \frac{\Delta p}{\Delta p_i} = \frac{C_p}{C_{p_i}}$$

Pressure Recovery Factor

Another common way of expressing pressure recovery is the ratio of static pressure at any section to the total pressure at the inlet.

$$\text{Pressure recovery factor PR} = \frac{p}{p_{\text{ot}}} \text{ (inlet)}$$

Viscosity and Temperatures Effects on Real Diffuser Performance

Ideally, the area ratio of the diffuser geometry is the only parameter of importance; however, viscous effects limit the

performance of real diffusers to less than ideal values, viscous effects reduce the pressure recovery and often lead to flow separation.

Diffusers employed for the MHD power generation system at UTSI have to operate under extreme conditions of temperature and slag carryover. This necessitates cooling of the diffuser walls and consequently establishes very high temperature gradients in the boundary layer. When the free stream temperature is much larger than the wall temperature, the thermophysical and transport properties are not constant in the boundary layer. A rigorous diffuser flow analysis should consider the variation in the properties of the fluid with temperature.

CHAPTER II

LITERATURE SURVEY

In this chapter, a brief survey of the relevant literature is presented. The material is presented in four sections, viz (i) subsonic diffuser performance and design criteria, (ii) models of velocity profile near walls, (iii) models of temperature profile near walls and (iv) turbulence transport models for the mainstream flows in the diffuser.

The material in each of these four sections must be taken into account when developing and evaluating a computational technology for predicting diffuser performance. The reason for examining these areas is that each comprises a part of the physics of the fluid mechanical processes taking place in the diffuser, both from a gross overview of overall system performance, and from a detailed consideration of the interactions of the fluid and the walls and the attendant internal mechanisms for transporting heat, mass, and momentum to (or from) the mainstream from (or to) the walls.

Subsonic Diffuser Performance and Design Criteria

S.J. Kline et al.(2) have formulated some criteria for the optimum design of straight walled diffusers. The two basic problems encountered in the design of straight walled diffusers are:

1. Prediction of overall flow regime.
2. Design of optimum straight walled diffusers.

Based on a literature survey they identified the following regimes of flow that occur in diffusers with one pair of diverging walls when the diffuser angle is continuously increased from zero, while the inlet flow conditions, the wall lengths and the throat width were held constant:

1. A region of no appreciable stall in which the main flow is well behaved and apparently unseparated.
2. A region of large transitory stall in which the separation varies in position, size, and intensity with time. This is a region of highly pulsating flows.
3. A region of fully developed stall in which the major portion of the diffuser is filled with a large recirculating region extending from the diffuser exit to a position close to the throat. The main stream attaches to and flows past one wall continuously and relatively smoothly.
4. A jet flow region in which the main flow is separated from both walls. The separation begins slightly downstream from the throat, and the flow does not reattach until well downstream from the diffuser. This occurs only at quite high angles of divergence.

Kline also states that the boundary layer profile apparently has relatively little effect on the flow regimes. The explanation for this result was apparently that the blockage was relatively small in the diffuser experiments that were studied. However, other work has shown that inlet boundary layer characteristics do have an effect on losses and recovery.

Runstadler and Dean (3), after careful studies on two dimensional diffusers, conclude that:

1. There is no critical inlet Mach number above which a precipitous decrease in diffuser pressure recovery occurs.

2. At low subsonic Mach numbers inlet or throat blockage is a critical parameter correlating diffuser performance. Increasing throat blockage results in decreasing pressure recovery, all other conditions being fixed.

3. Aspect ratio is an important as other geometric parameters of the diffuser in establishing pressure recovery. For fixed geometry and blockage, maximum pressure recovery occurs at low aspect ratio.

Many of the aerodynamic and geometric parameters which influence the performance of diffusers have been reviewed by Reneau, et al.(4). They presented the effects of the following parameters on the performance of the diffuser:

1. Aspect ratio.
2. Cross-Sectional shape.
3. Reynold's number.
4. Subsonic inlet Mach number.
5. Inlet turbulence.
6. Inlet velocity profile distortion.
7. Tail pipes.
8. Wall fairing.

Because these results are important, they are discussed below in more detail.

Aspect Ratio Effects

For aspect ratio, greater than unity aspect ratio does not appreciably affect the flow regimes identified by Kline, et al.; however, inlet aspect ratio does affect recovery. Increases in aspect ratio above $AS = 1$ raise recovery slightly, but decreases in the aspect ratio below $AS = 1$ reduce recovery severely, especially for diffusers for small angle 2θ . The recovery changes very little for $AS > 10$. For small aspect ratio, especially for $AS < 1$, diffusers with small divergence angles have much lower recovery than those with larger divergence angles.

The exit plane boundary layer displacement thickness is smaller in a smaller aspect ratio than in an otherwise identical large aspect ratio diffuser with identical inlet conditions. However, the reduction in displacement thickness at the exit does not improve performance because, as the aspect ratio is reduced, the ratio of diffuser total exit area to exit area effectively blocked by the boundary layer also decreases. Hence, performance always decreases with decreasing aspect ratio.

Cross-Sectional Shape Effects

The geometric parameters chosen to describe the diffuser geometry have a large effect on the comparisons of performance between various cross-sectional shaped diffusers. The area ratios of the three diffusers types are given by the following equations:

$$\text{Two dimensional (plane) diffusers: } AR = 1 + \frac{N}{W_1} 2 \tan\theta \quad (2.1)$$

$$\text{Square: } AR = 1 + \frac{2N}{W_1} 2 \tan\theta + \left(\frac{N_2}{W_1}\right)^2 (\tan\theta)^2 \quad (2.2)$$

$$\text{Conical: } AR = 1 + \frac{2N}{D_1} 2 \tan\theta + \left(\frac{N_2}{D_1}\right)^2 (\tan\theta)^2 \quad (2.3)$$

The similarity length variables are defined as N/W_1 , $2N/W_1$, and $2N/D_1$ for two dimensional square and conical diffusers respectively. Reneau, et al. (4) states the following conclusions:

1. Depending on the geometrical parameters chosen to describe the geometry, either the two dimensional or square diffuser geometry may give higher recovery. Conical diffusers give as high as or higher recovery than two dimensional diffusers.
2. The peak recovery for conical diffusers occurs at essentially the same location when presented in AR and similarity length coordinates.

Reynold's Number Effects

The Reynold's number described here is based on the inlet conditions $Re = \bar{U}_1 W_1 / \nu$. For subsonic diffusers when the inlet Mach number is increased by increasing the velocity, Reynold's number also increases. The recovery drops rapidly once the flow becomes locally supersonic anywhere in the diffuser. This generally occurs when Mach number based on average inlet velocity is about 0.8. Below

peak recovery, recovery is independent of Reynold's number for $Re > 5.0 \times 10^4$, at lower Reynold's numbers, the performance drops off slightly.

Inlet Mach Number Effects

The tests of Young and Green (19) show that the recovery is essentially constant with changes in Mach number until the flow becomes locally supersonic going around the corner at the throat. Then recovery drops drastically with very small increase in Mach number. For Mach numbers below $M = 0.7$, the recovery is practically independent of Mach number.

Inlet Turbulence

High inlet turbulence will increase the peak recovery. For very high turbulence intensity, the performance in the large transitory stall regions is greatly increased and the onset of two-dimensional stall is delayed. It can be concluded that this effect is similar to the effect of turbulence intensity which delays flow separation from bodies immersed in a fluid stream.

Inlet Velocity Profile Distortion

Thick inlet boundary layers reduce the pressure recovery. Also, when the inlet velocity profile is skewed or non-uniform, the performance is decreased and the divisions between the flow regime occur at lower values of θ than for symmetric inlet flow. The magnitude of pressure recovery changes were not adequately correlated to inlet profiles because the general problem depends on other factors such as the level of inlet turbulence, etc.

Tail Pipes

If a constant area duct (a tail pipe) follows the diffuser, there is a pressure rise in the part of the duct as the profiles becomes more uniform tending toward pipe flow profiles.

However, the performance of the diffuser itself is independent of the presence or lack of tail pipe. The fraction of the pressure rise occurring in the tail pipe is very small for diffuser geometries below peak recovery, but it increases as the amount of stall in the diffuser increases.

Wall Fairing

Diffusers must often be designed with given length and area ratio. When the simple two dimensional straight-wall diffuser which fits the given geometry has poor performance or an undesirable flow pattern, there are several possible mechanical remedies; one of these is wall fairing, wherein the diffuser wall is contoured to control the local pressure gradient. Two types of walls are commonly used. The convex walls results from attempts to obtain maximum pressure rise in a given length, while the concave wall results from trying to avoid stall and thereby attain high performance. The concave shape allows a significant amount of diffusion before sharp increase in divergence at which separation occurs. The two types of wall shapes are given in Figure 4.

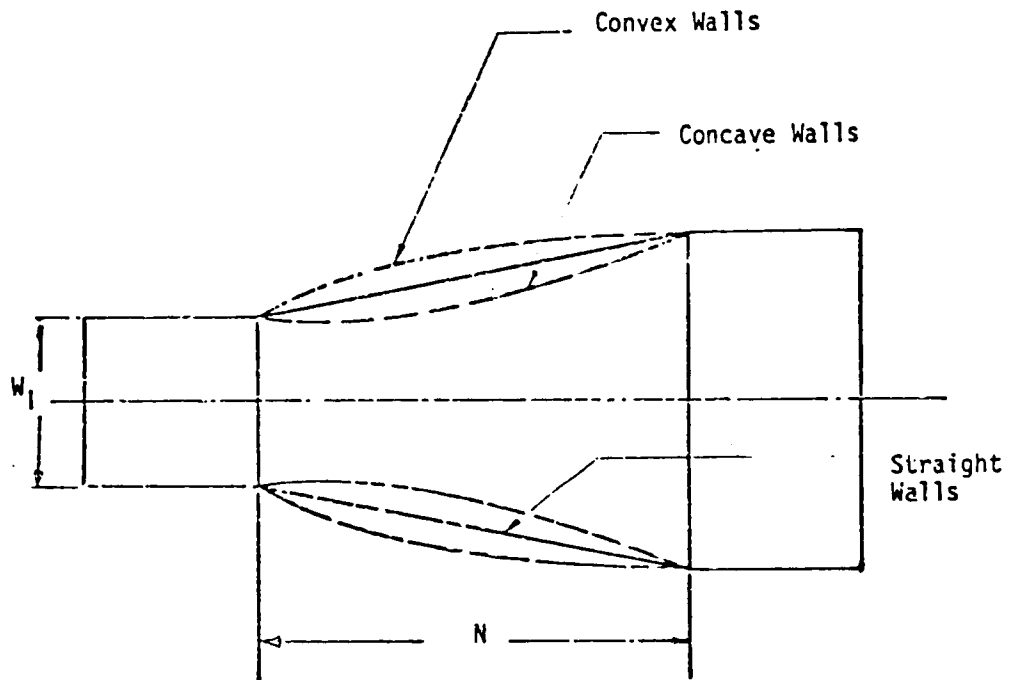


Figure 4. Wall Fairing

Velocity Distribution in MHD Diffusers

The velocity profiles emerging from an MHD generator channel are unique. The MHD channel flows are acted upon by the $J \times B$ Lorentz force in addition to pressure and viscous forces. Since the $J \times B$ is a force that retards the motion of the gas and since large temperature gradients exist in the boundary layer adjacent to the walls, the flow is typically lower in velocity in the center of the free stream, such that the peak velocity shifts from the centerline toward the walls. This phenomenon is called velocity overshoot and has been studied theoretically by Maxwell, et al.(5) and experimentally verified by Rankin, et al.(6). Greywall and Smith (7) have studied the effect of velocity overshoot on the performance of subsonic MHD diffusers. A comparison between velocity overshoot and fully developed profiles has indicated that the velocity overshoot condition generates significantly greater wall shear stress and hence, should be less susceptible to flow separation than the fully developed profile. Also, the heat flux associated with the overshoot condition is higher and has serious effects upon the overall heat transfer design of the diffuser. They have also shown that the velocity overshoot condition is amplified by diffuser divergence angles greater than some critical value because of pressure gradient effects. It should be noted that the study was based on air (ideal gas) as the working fluid and a mathematically convenient velocity overshoot model.

Therefore, a numerical method for predicting diffuser performance for MHD applications must be able to describe non-uniform diffuser velocity inlet profiles of the overshoot variety. And, the method must be able to accurately describe the interaction of the flow with the walls. Hence, the next section provides a brief account of the nature of velocity profiles in turbulent flow near walls.

Boundary Layer Velocity Profiles

The previous sections discussed the performance of subsonic diffusers for MHD applications as affected by boundary layer flow in an adverse pressure gradient, and over very rough walls. The families of velocity profiles from turbulent flow over walls and in pipes is described in this section. To provide an appropriate background for this physical phenomenon, a general description of turbulent boundary layers is provided.

The concept of boundary layer is that there is a layer of fluid adjacent to solid surface in which viscous forces are of the same order of magnitude as inertia forces. A fundamental assumption of boundary layer theory is that the fluid in contact with the body surface is at rest relative to the body. Thus, the hydrodynamic or the momentum boundary layer may be defined as the region in which the fluid velocity changes from its free stream value to zero at the body surface. The flow in the wall region is determined by the conditions at the wall as expressed by the wall shear stress and its roughness.

The roughness can be characterized by one length parameter d , for instance, the average height of the roughness elements. The mean velocity component in this region, then, will be a function of wall shear stress τ_w , roughness parameter d , the distance, y , to the wall and the fluid properties μ and ρ . From the equations of motion, it becomes evident that the ratio τ_w/ρ is the quantity that correlates the velocity distributions. Since the ratio has the dimension of velocity squared, a characteristic velocity called the wall-friction velocity has been defined as

$$u_\tau \equiv \text{wall-friction velocity} = \left(\frac{\tau_w}{\rho} \right)^{1/2}$$

The following non-dimensional y -coordinate has also been defined

$$y^+ \equiv \frac{y u_\tau}{\nu}$$

The expression for velocity profile near the wall can be then written as

$$\frac{u}{u_\tau} = f \left(\frac{u_\tau y}{\nu}, \frac{u_\tau d}{\nu} \right) \quad (2.4)$$

Experimental evidence has shown that the general boundary layer can be described as having three regimes called:

the inner layer (laminar shear dominates)

the logarithmic layer (turbulent shear dominates)

the wake regime (fully turbulent)

In the inner layer experiment shows that

$$u^+ = y^+ \quad (2.5)$$

For the logarithmic layer the velocity profiles behaves as

$$u^+ = \frac{1}{k} \log_e (y^+) + B \quad (2.6)$$

where k and B are dimensionless constants.

Note that Spalding (8) gave the following wall velocity profile expression

$$y^+ = u^+ + e^{-kB} \left[e^{ku^+} - 1 - ku^+ - \frac{(ku^+)^2}{2!} - \frac{(ku^+)^3}{3!} - \frac{(ku^+)^4}{4!} \right] \quad (2.7)$$

which, for a relatively smooth wall ($k = 0.4$ and $B = 5.5$), describes the profile from wall to the outer edge of the logarithmic layer.

The wake law describes the wake regime of the boundary layer and is sensitive to the pressure gradient parameter. The outer or wake regime has been found to obey a profile of the following form

$$U - u = f(\tau_w, \rho, y, \delta, \frac{dp}{dx}) \quad (2.8)$$

where U is the freestream or maximum velocity. After nondimensionalization, the outer law assumes the form

$$\frac{U-u}{u_\tau} = g(y/\delta, \delta/\tau_w \frac{dp}{dx}) \quad (2.9)$$

Taking δ as the displacement thickness δ^* , Clauser (8) defined an "equilibrium parameter" β , as

$$\beta \equiv \frac{\delta^*}{\tau_w} \frac{dp}{dx} \quad (2.10)$$

Clauser (3) then showed that a boundary layer with variable pressure gradient but $\beta = \text{constant}$ is in turbulent equilibrium in the following sense. All the gross properties of the boundary layer can be scaled with a single parameter, the defect thickness Δ defined as

$$\Delta = \int_0^{\infty} \frac{U-u}{u_{\tau}} dy = \delta^* \lambda \quad (2.11)$$

$$\text{where } \lambda = (2/C_f)^{1/2}$$

Coles (3) also has proposed the following curve fit approximation for the outer or wake regime by defining the so-called law-of-the-wake.

$$W(y/\delta) \approx 2 \sin^2 \left(\frac{\pi}{2} \frac{y}{\delta} \right) \quad (2.12)$$

Then, using $W(y/\delta)$, the law-of-the-wall, law-of-the-wake is constructed which is an effective composite function to describe both the logarithmic and wake regimes. It is

$$u^+ = \frac{1}{k} \log_e (y^+) + \frac{\pi}{k} W (y/\delta) \quad (2.13)$$

where π is called the wake parameter and has a value of 0.55 for nearly zero pressure gradient flows. Granville (9) presented a simple polynomial for the wake function which was expressed in terms of Coles parameter π

$$g (\tilde{\pi}, y/\delta) = \frac{1}{k} (1+6\tilde{\pi}) (y/\delta)^3 - \frac{1}{k} (1+4\tilde{\pi}) (y/\delta)^3 \quad (2.14)$$

It has been found that a cubic equation in (y/δ) provides a good fit to the mean velocity profile for turbulent boundary layers that develop from uniform initial flows subjected to "mild" pressure gradients.

Other expressions for profile families have been developed in the past. For example, Rasmussen (10) has presented an expression for the inner layer given by

$$y^+ = u^+ + e^{-kB} [2 \cosh (ku^+) - (ku^+)^2 - 2] \quad (2.15)$$

From the formulas by Spalding and by Rasmussen, R.B. Dean (11) has derived a single analytical function

$$y + e^{kg} (\pi, y/\delta) = u^+ \quad (2.16)$$

for the complete mean velocity profile in a turbulent boundary layer.

R.A. McD Galbraith, et al.(12) has shown that the law-of-the-wall is universal in the sense that it has a wide range of validity even in flows with large pressure gradients, provided the boundary layer is of a sufficiently high Reynold's number.

They observed that in sufficiently severe adverse pressure gradients, the universal law-of-the-wall breaks down, but that the mechanism of breakdown is extremely important in its implications. Theoretically, the breakdown could either be a general breakdown or a progressive breakdown of the logarithmic profile, as shown in Figure 5. In either case, departures from the log law will become noticeable progressively closer to the wall with increasingly adverse

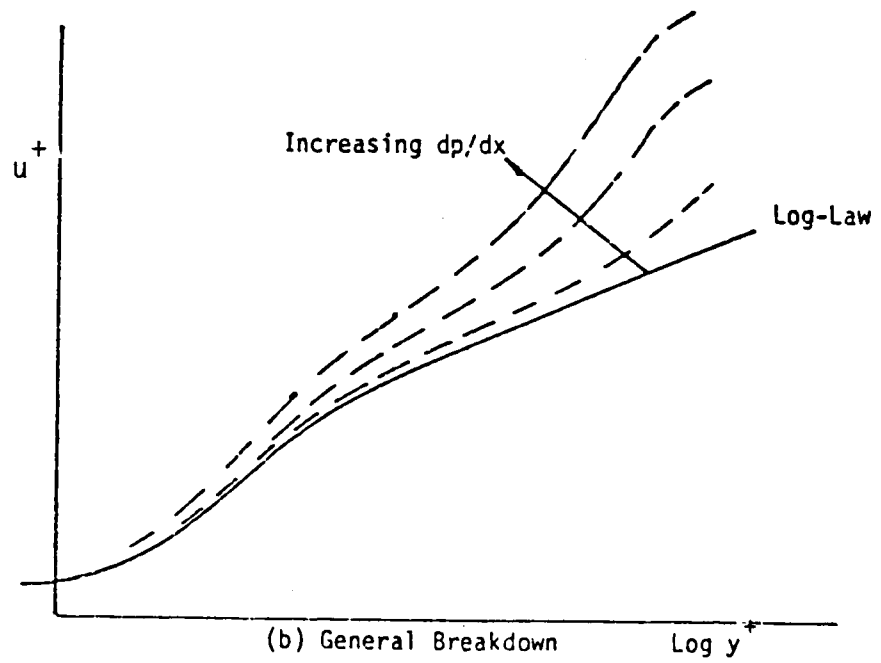
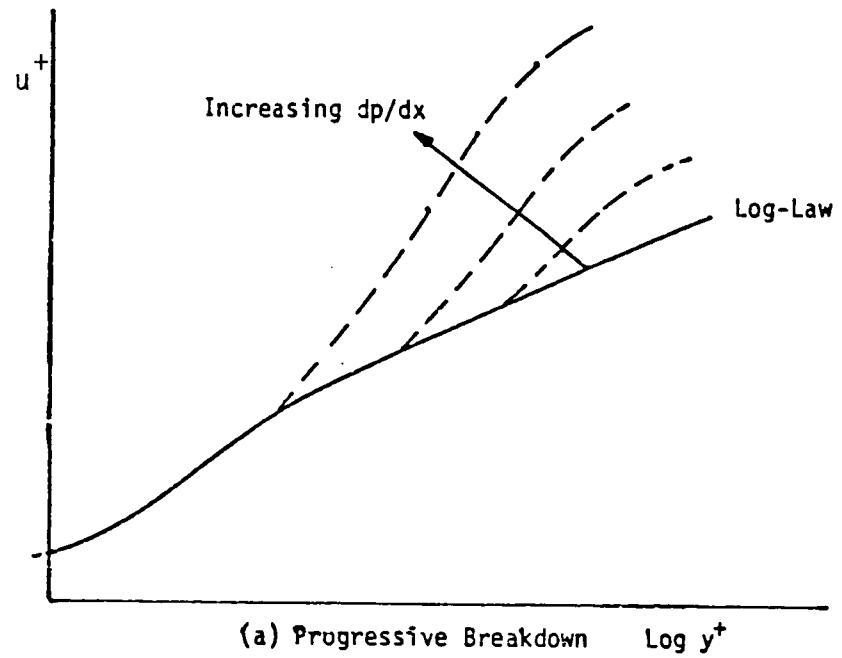


Figure 5. Possible Modes of Log-Law Breakdown in Adverse Pressure Gradients

pressure gradient, but the distinction between the two modes of breakdown is clear. These authors demonstrate that the experimental evidence supports the progressive breakdown concept for high Reynold's number flows.

On the other hand, the observations of McD Galbraith et al. must be compared to those of Launder [Figure 4.2, Reference 13] which shows that low Reynold's number boundary layers breakdown generally. Hence for low Reynold's number boundary layers, the profiles of velocity deviate strongly from the law-of-the-wall shape and tend toward the laminar (or linear) velocity profile. There seems to be evidence, therefore, that both the progressive and general breakdown modes exist in turbulent boundary layers. For high Reynold's numbers, the breakdown is progressive under increasingly adverse pressure gradients. For low Reynold's numbers, the breakdown is general as the Reynold's number of flow decreases. Clearly then there will exist a class of low Reynold's number boundary layers that exhibit characteristics of both general and progressive breakdown from the law-of-the-wall as adverse pressure gradients become stronger.

The boundary layer flows studied in the present research are known to be characterized by high Reynold's number conditions. Hence, the present study is limited to boundary layers wherein the law-of-the-wall behavior of the velocity can be assured in at least some part of the layer.

For high Reynold's number case, McD Galbraith et al.(12) has shown that the parameter

$$\frac{\tau}{\tau_w} = 1 + p^+ y^+ + \frac{1}{2} a^+ \{C_1 + y^+ (C_2 (\log_e y^+)^2 + C_3 \log_e y^+ + C_4)\} \quad (2.17)$$

$$\text{where } p^+ = \frac{v}{\rho u_\tau^3} \frac{dp}{dx}$$

$$y^+ = \frac{u_\tau y}{v}$$

$$a^+ = \frac{v}{\rho u_\tau^3} \frac{d\tau_w}{dx} \quad \text{and}$$

$$C_1 = -777.0; C_2 = 5.706; C_3 = 14.62 \text{ and } C_4 = 15.08$$

can be used to define an effective law-of-the-wall for severe adverse pressure gradient flows.

Velocity Profile on Rough Walls

The general law-of-the-wall expression can be written as

$$\frac{u}{u_\tau} = f\left(\frac{u_\tau y}{v}, \frac{u_\tau d}{v}\right) \quad (2.18)$$

where d is a characteristic roughness length parameter, for example, the average height of the roughness elements. For a smooth wall $d = 0$. For the turbulent boundary layer along a very rough wall, the average height of roughness elements is often greater than the thickness of the inner layer, hence the effect of laminar viscosity should

disappear for rough walls. Hence, the expression for the velocity distribution should not involve the molecular viscosity of the fluid. Elimination of the kinematic viscosity results in the following law-of-the-wall expression valid for "completely rough" walls:

$$u^+ = \frac{1}{k} \log_e (y/d) + \text{Constant} \quad (2.19)$$

The outer region of the boundary layer is then given by a velocity defect law in the form:

$$\frac{u-u_\tau}{u_\tau} = \Psi (y/\delta) \quad (2.20)$$

The velocity defect or wake law applies to smooth and rough walls. There is an uncertainty in the estimation of the value of y where the velocity is, on the average, zero along the rough wall. The point where $u=0$ may be somewhere between $y=0$ and $y=d$. As long as the thickness of the viscous sublayer is greater than d , the expected derivation of the velocity distribution from that valid for a smooth wall will be small.

To show how the roughness affects the velocity profile, subtract equations 2.19 from equation 2.6 and get

$$\frac{\Delta u}{u_\tau} = u_{\text{Smooth Wall}}^+ - u_{\text{Rough Wall}}^+ = \frac{1}{k} \log_e \left(\frac{d}{y} \right) + \text{Constant} \quad (2.21)$$

Thus, the rough wall velocity profile shifts downward from the classic law-of-the-wall, when plotted in u^+ - y^+ co-ordinates by an amount ΔB which depends on the dimensionless roughness $d^+ = \frac{d u_\tau}{\nu}$. The classical law-of-the-wall expression can be rewritten in the form which includes both smooth and rough wall effects as follows:

$$u^+ = \frac{1}{K} \log_e (y^+) + 5.5 - \Delta B(d^+) \quad (2.22)$$

Unfortunately, the experimental evidence is that ΔB is not a unique function of d^+ but varies with the type of roughness. For $d^+ > 60$, ΔB can be approximated by the following expression

$$\Delta B = \frac{1}{K} \log_e(d^+) - 3.0 \quad (2.23)$$

For sand grain roughness the following regimes can be identified.

$d^+ > 4$ hydraulically smooth wall

$4 < d^+ < 60$ transitional rough regime

$d^+ > 60$ fully rough flow

The approximate additive constant in the law-of-the-wall expression takes the following form

$$B \approx 5.5 - \frac{1}{K} \log_e (1 + 0.3 d^+) \quad (2.24)$$

Pimenta, et al.(14) have carried out experiments on rough wall boundary layers and have shown that the mean velocity profile agreed well with the predicted form:

$$u^+ = \frac{1}{0.41} \ln (y/d) + 8.5 \quad (2.25)$$

Lewis (15) has presented a unified approach by making use of an analysis which is based upon an approximate model of separated flow over each roughness element. The analysis was carried out for rectangular rib roughness with equally spaced rectangular ribs. Other shapes were approximated by a form drag coefficient and an equivalent rectangular rib geometry.

Perry, et al.(16) have conducted detailed experimental studies of turbulent boundary layer development over rough walls in both zero and adverse pressure gradients. Two wall roughness geometries were chosen, each giving a different law of behavior. One type gives a roughness function which depends on Reynold's number based on the shear velocity and on a length associated with the size of roughness. The other type of roughness is typified by a smooth wall containing a pattern of narrow cavities. They conclude that the latter type of roughness does not follow the Nikuradse-Clauser correlation scheme discussed earlier. For fully rough flows, the roughness functions for both types of roughness functions of a length scale E . They expressed the relationship of

$$\frac{\Delta u}{u_\tau} = \frac{1}{k} \log_e \left(\frac{Eu_\tau}{\nu} \right) + \text{Constant} \quad (2.26)$$

E can be easily scaled with respect to the size of the roughness elements for some types of roughness, but there is no straight forward way of predicting E for general roughness elements.

Thermal Boundary Layers

The thermal boundary layer is the region of flow next to the surface of a body immersed in the fluid, or a surface confining the fluid wherein the temperature varies from the fluid temperature to the surface temperature. In general, the thermal boundary layer is a

consequence of the motion of the fluid over the surface, the surface condition (i.e., whether rough or smooth) and certain dimensionless parameters including Reynold's number, Prandtl number, Eckert number, and Mach number. Like the velocity or hydrodynamic boundary layer, thermal boundary layer has been determined by experiment to exhibit characteristic temperature profile shapes for a wide range of turbulent flows. It seems reasonable, based on observations and supportive theoretical considerations, that the wall matching technique used to predict the velocity field boundary conditions for numerical solution procedures can be extended to provide temperature field boundary conditions for numerical solutions. To carry out, this requires that a thermal law-of-the-wall be selected or developed from the literature capable of including the effects of Reynold's number, Prandtl number, and roughness characteristics.

Reynold's Analogy

One of the earliest attempts to describe the thermal boundary layer as a law-of-the-wall profile was that of Von Karman (1939), who used what is called Reynold's analogy. This method assumes that the turbulent momentum diffusivity is equal to the turbulent energy diffusivity or

$$\epsilon_h = \epsilon_m \text{ or}$$

equivalently a turbulent Prandtl number can be defined as

$$Pr_t = \frac{\epsilon_m}{\epsilon_h}$$

and $Pr_t = 1.0$ for the turbulent regime of the boundary layer. On this basis, Von Karman derived a family of temperature profiles for the thermal boundary layer which are directly related to the velocity profiles. The fundamental relationship being expressed by

$$du = \left(\frac{\tau_w}{\rho} \right) \frac{\rho C_p}{q_w} d\theta \quad (2.27)$$

where $\theta = T_w - T_\infty$ is the difference between wall and freestream fluid temperature. Hence for a scaled temperature,

$$t^+ = \left(\frac{\tau_w}{\rho} \right)^{1/2} \frac{\rho C_p}{q_w} \theta$$

the relationship becomes

$$du^+ = dt^+ \quad (2.28)$$

or

$$t^+ = u^+ + \text{Constant}$$

Hence, the shape of the temperature profile and the shape of the velocity profile are the same in the turbulent regime of the fluid. The theory applies best where the molecular and the turbulent Prandtl numbers are close to unity.

Experiment has confirmed such behaviors for many cases. The general problem is the range of validity for such thermal boundary layer profiles for flows with nonunity Prandtl numbers, and for flows affected by surface roughness. What follows is a brief presentation of typical profile families that theory and experiment have provided and are found in literature.

Thermal Boundary Layer Profile Functions

The boundary layer energy equation can be written in the form

$$u \frac{\partial t}{\partial x} + v \frac{\partial t}{\partial y} + \frac{1}{\rho C_p} \frac{\partial q''}{\partial y} = 0 \quad (2.29)$$

where $q'' = -\rho C_p (\alpha + \epsilon_h) \frac{\partial t}{\partial y}$ = the total heat flux.

The following law-of-the-wall expression has been derived from the energy equation

$$t^+ = 13.2 \text{ Pr} + \frac{\text{Pr} t}{k} \log_e (y^+/13.2) \quad (2.30)$$

provided by Reference (16).

A comparison of this law-of-the-wall expression, with experiment, is given in Figure 6 (17), and it shows that the viscous sublayer extends to at least $y^+ = 30.0$.

For a hydraulically rough surface a law-of-the-wall expression was given in (17) as

$$t^+ = \frac{1}{\text{St}_d} + \frac{\text{Pr} t}{k} \log_e \left(\frac{32.6 y^+}{\text{Re}_d} \right) \quad (2.31)$$

where St_d = Stanton number based on u_τ and Re_d = Reynold's number based on roughness. St_d must be determined from experiment and it is a function of the type of surface roughness present. Dipprey, et al. (18) have given the following correlation expression for St_d .

$$\text{St}_d = C \text{Re}_d^{-0.2} \text{Pr}^{-0.44} \quad (2.32)$$

where C is again a constant dependent on the type of roughness.

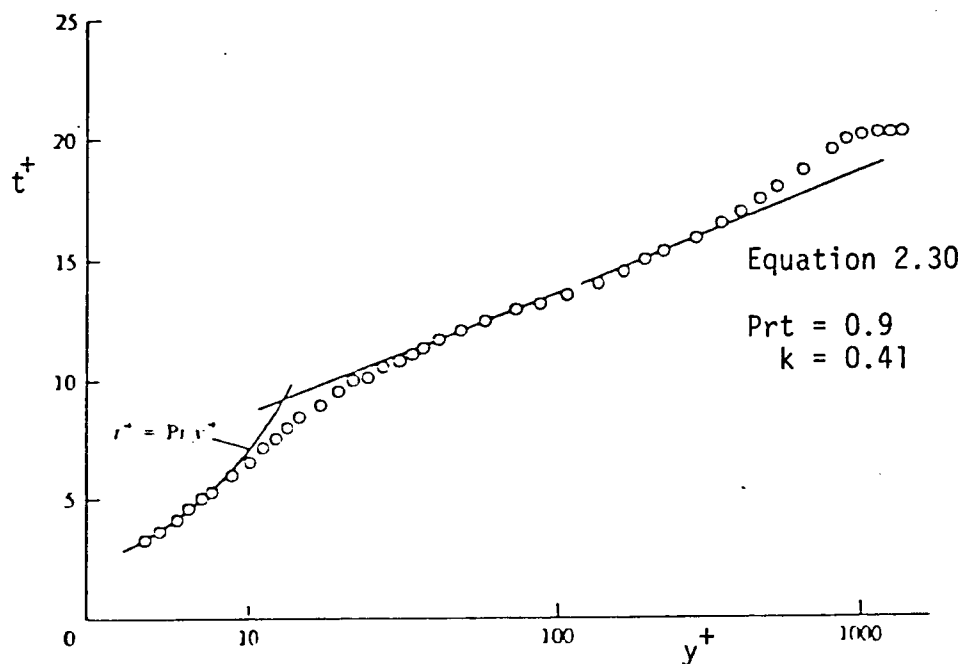


Figure 6. Temperature Profile for Flow of Air ($Pr=0.7$)
Along a Plate with Constant Free Stream
Velocity and Constant Surface Temperature

Typical values for C are ≈ 1.00 . For circular tubes with fully developed velocity and temperature profiles and with constant heat rate, the law-of-the-wall was given by Reference (25) as

$$t^+ = 13.2 \text{ Pr} + \frac{\text{Pr}_t}{k} \log_e \left\{ y^+ \left(\frac{1.5 (1 + r/r_0)}{1 + 2(r/r_0)^2} \right) \right\} \quad (2.33)$$

J. Blom (19), concludes that for developing boundary layers, based on experimental results,

1. The usually assumed Reynold's analogy ($\text{Pr}_t = 1.0$) is invalid. Pr_t varies across the boundary layer.

2. Close to the wall, ($y^+ < 30$) a fairly uniform distribution of Pr_t exists, giving increasing values of Pr_t with increasing distance from the wall.

As a last expression, the work at the Imperial College of London in the Mechanical Engineering Department suggested a law-of-the-wall for thermal boundary layers that was presented by Launder and Spalding, Reference (20), and is

$$\begin{aligned} \frac{(t_p - t_w) C_p \rho C_u^{1/4} K_p^{1/2}}{q_w} &= \frac{\text{Pr}_t}{k} \log_e \left\{ \frac{E y_p}{\nu} (C_u^{1/2} K_p)^{1/2} \right\} \\ &+ \text{Pr}_t \frac{\pi/4}{\sin(\pi/4)} \left(\frac{A_t}{k} \right)^{1/2} \left(\frac{\text{Pr}}{\text{Pr}_t} - 1 \right) \left(\frac{\text{Pr}_t}{\text{Pr}} \right)^{1/4} \end{aligned} \quad (2.34)$$

where:

t_p = Temperature at a point denoted p off the wall in the turbulent layer

C_μ = A constant equal to approximately 0.09

K_p = The turbulent kinetic energy at the point p

Pr_t = The turbulent Prandtl number

Pr = Laminar or fluid Prandtl number

E = A "constant" that is an apparent function of the wall roughness, approximately equal to 9.0 for a smooth wall

A_t = A constant called "Van Drest's constant" whose value is equal to 26.0 for a smooth wall

C_p = The constant-pressure specific heat

Note that from the paper by Launder and Spalding, (20)

$$\frac{\tau}{\rho} = C_\mu^{1/2} K = \text{constant} \quad (2.35)$$

Launder and Spalding claim that the expression for the wall thermal profile is flexible enough to account for wall roughness, pressure gradient, and mass transfer through the wall by suitable modifications (or correlations) of the "constants" E and A .

Turbulence Transport Model

The model used in the present study is a high turbulent Reynold's number, one-equation model. The model was developed by Lee and Harsha, Reference (21) and Harsha, Reference (21), and was applied to non-separating diffuser flows by Harsha and Glassman, Reference (1). The model is based on the relationship between shear stress and

turbulent kinetic energy that Bradshaw observed for boundary layer flows and Harsha and Lee noted in free shear layers. That is, that

$$\tau_t = a_1 \rho K \quad (2.36)$$

where a_1 is a "constant" whose value is about 0.3. The strengths and weaknesses of this model are reported elsewhere, for example, Reference (23). However, for many different types of turbulent flows, it has proven adequate. In the present application, it was selected because of the diffuser calculations reported by Harsha and Glassman, Reference (1). In their paper these authors show fairly good agreement between experimental and predicted skin friction coefficients and pressure distributions but the mean axial velocity profiles were not well predicted. In the present study, an attempt was made to overcome this deficiency.

The model is based on a single transport equation for K with a prescribed length scale. The differential transport equation is

$$\begin{aligned} \rho u \frac{\partial K}{\partial x} + \rho v \frac{\partial K}{\partial r} &= \frac{1}{r} \frac{\partial}{\partial r} \left(\left(\frac{\mu + \mu_t}{Pr_k} \right) r \frac{\partial K}{\partial r} \right) \\ &+ \mu_t \left(\frac{\partial u}{\partial r} \right)^2 - \frac{a_2 \rho K^{3/2}}{l_k} \end{aligned} \quad (2.37)$$

The left hand side of equation 2.37 expresses the convection of K by the time-mean motion while the right hand side of the equation expresses diffusion, production and dissipation, respectively, of the turbulent kinetic energy.

The length scale l_k , requires some explanation. The dissipation term represents isotropic turbulence dissipation rate, as explained by Hinze, Reference (24). However, the length scale usually used in the model is a macroscopic length scale selected based on some physically perceivable scale. For example, for free jets, Harsha found that l_k could be set proportional to the local jet width or half-width at each x or streamwise station. For this class of flows, with the value of l_k set of 0.98, Harsha found that the constant a_2 had to be about 1.69 for optimum prediction success. However, when applying this model to boundary layer flows, the length scale had to be selected proportional to the Prandtl mixing length. For boundary layer flows, with l_k set equal to the Prandtl mixing length distribution, the necessary value of the constant a_2 was found to be 0.169. Harsha and Glassman decided to keep the value of a_2 constant for both boundary layer flows and free jet type flows, i.e., by setting $a_2=1.69$. Then, for boundary layer flows, they scaled l_k up by multiplying the Prandtl mixing length distribution by 10.0. Finally, they determined, by numerical experimentation and optimization, a distribution for l_k that is somewhat different from Prandtl Mixing Length. The distribution is shown in Figure 7.

In the present study, the length-scale distribution was again modified to take into account pressure gradient effects. The results of the length-scale modification will be shown later.

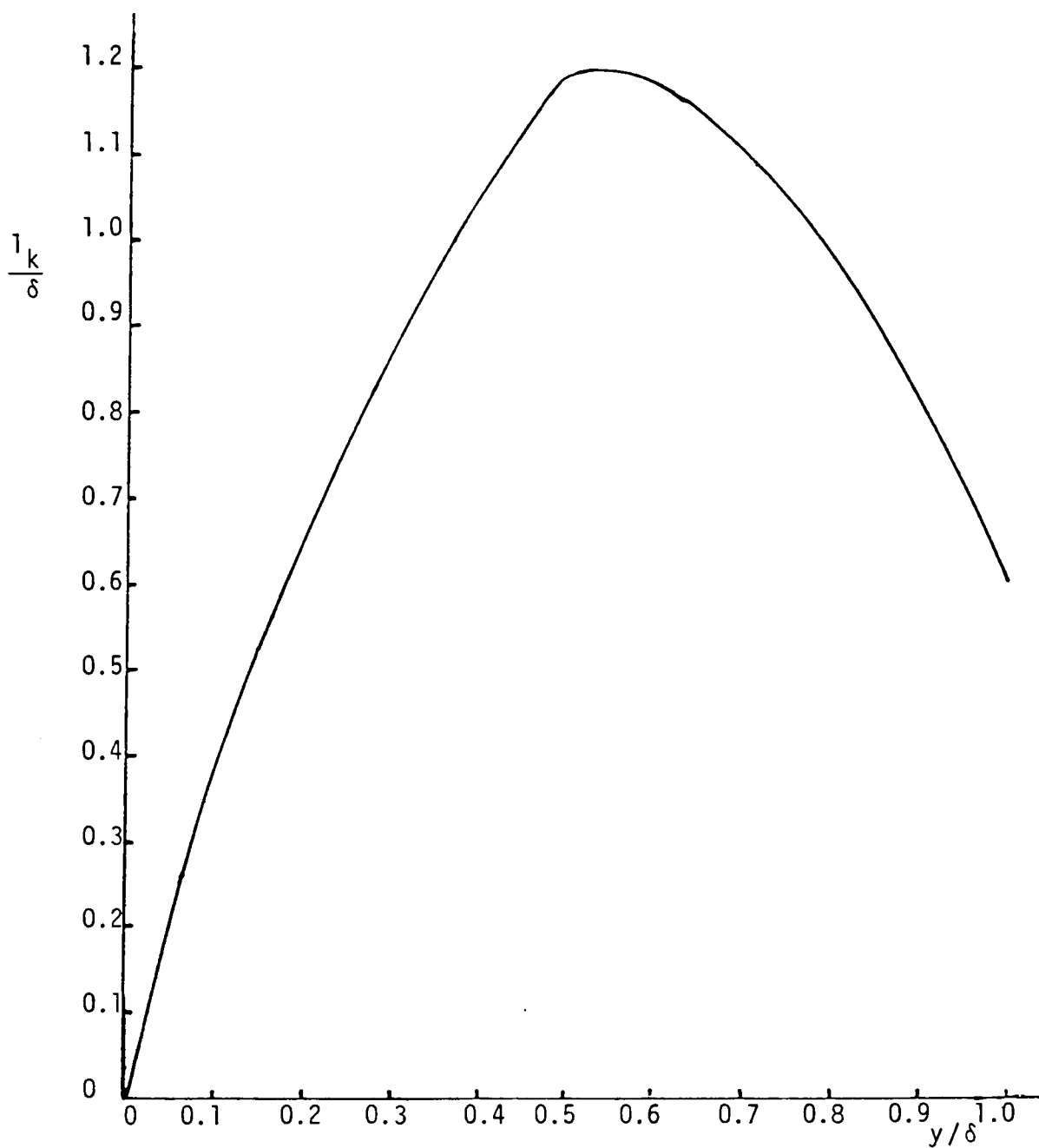


Figure 7. Length Scale Distribution in Diffuser Boundary Layer
Assumed by Harsha and Glassman (1)

CHAPTER III

THEORY AND NUMERICAL SOLUTION PROCEDURE

The analysis presented in this thesis is based on the numerical frame-work of GENMIX program developed by Patankar and Spalding (25). The present analysis uses a one-equation turbulent kinetic energy (TKE) model for evaluating turbulent shear stress, with a prescribed length scale distribution for the dissipation term, and is an extension of the work of Harsha and Glassman (1).

Governing Equations

The co-ordinate system employed for the derivation of boundary layer equations is given in Figure 8. The conservation equations for the boundary layer flow are given below in the form favored by Patankar and Spalding. Conservation of momentum in the x-direction

$$\rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial r} = - \frac{dp}{dx} + \frac{1}{r} \frac{\partial}{\partial r} (\tau r) \quad (3.1)$$

Conservation of stagnation enthalpy

$$\rho u \frac{\partial \tilde{h}}{\partial x} + \rho v \frac{\partial \tilde{h}}{\partial r} = \frac{1}{r} \frac{\partial}{\partial r} [(J_h - u\tau)r] \quad (3.2)$$

Turbulent kinetic energy equation

$$\rho u \frac{\partial K}{\partial x} + \rho v \frac{\partial K}{\partial r} = \frac{1}{r} \frac{\partial}{\partial r} \left\{ \left(\frac{\mu + \mu_t}{Pr_k} \right) r \frac{\partial K}{\partial r} \right\} + \mu_t \left(\frac{\partial u}{\partial r} \right)^2 - \frac{a_2 \rho k^{3/2}}{l_k} \quad (3.3)$$

Conservation of mass

$$\frac{\partial}{\partial x} (\rho u r) + \frac{\partial}{\partial r} (\rho v r) = 0 \quad (3.4)$$

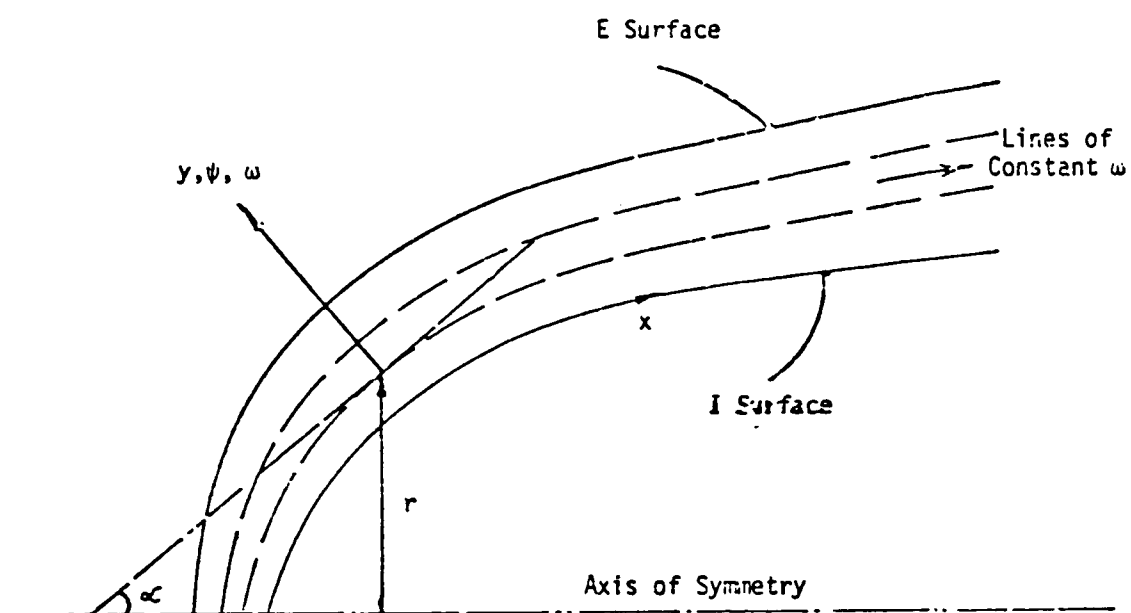


Figure 8. Coordinate System

The conservation equations are transformed from the x-y plane to x- ω plane. The transformed equations can be written as follows:

Momentum equation

$$\frac{\partial u}{\partial x} + \frac{\partial u}{\partial \omega} \left\{ \frac{r_{I\dot{m}_I} + \omega [r_{E\dot{m}_E} - r_{I\dot{m}_I}]}{(\Psi_E - \Psi_I)} \right\} = \frac{\partial}{\partial \omega} \left\{ r \mu_{eff} \frac{\partial u}{\partial \omega} \frac{1}{(\Psi_E - \Psi_I)^2} \rho u \right\} - \frac{1}{\rho u} \frac{dp}{dx} \quad (3.5)$$

Total enthalpy equation

$$\frac{\partial \tilde{h}}{\partial x} + \left\{ \frac{r_{I\dot{m}_I} + \omega (r_{E\dot{m}_E} - r_{I\dot{m}_I})}{(\Psi_E - \Psi_I)} \right\} \frac{\partial \tilde{h}}{\partial \omega} = \quad (3.6)$$

$$\frac{\partial}{\partial \omega} \left\{ \frac{r^2 \rho u \mu_{eff}}{(\Psi_E - \Psi_I)^2} \frac{\partial \tilde{h}}{\partial \omega} \right\} + \frac{\partial}{\partial \omega} \left\{ \frac{r^2 \rho u \mu_{eff}}{(\Psi_E - \Psi_I)^2} \left(1 - \frac{1}{Pr_{eff}} \right) \frac{\partial}{\partial \omega} \left(\frac{u^2}{2} \right) \right\}$$

Turbulent kinetic energy equation

$$\frac{\partial K}{\partial x} + \frac{\partial K}{\partial \omega} \left\{ \frac{r_{I\dot{m}_I} - \omega (-r_{E\dot{m}_E} - r_{I\dot{m}_I})}{(\Psi_E - \Psi_I)} \right\} = \frac{\partial}{\partial \omega} \left\{ \frac{(\rho u)^2 \varepsilon u}{Pr_K (\Psi_E - \Psi_I)^2} \frac{\partial K}{\partial \omega} \right\} + \frac{\tau r}{(\Psi_E - \Psi_I)} \frac{\partial u}{\partial \omega} - \frac{a_2 \rho K^{3/2}}{1_k \rho u} \quad (3.7)$$

where Ψ = stream function $\Psi(x, y)$

$\omega \equiv$ non-dimensional stream function $\frac{\Psi - \Psi_I}{\Psi_E - \Psi_I}$

where Ψ_I and Ψ_E are the values of stream functions at inner and outer boundaries.

From the definition of Ψ , we have

$$\frac{d\psi_I}{dx} = -r_I \dot{m}_I''$$

$$\frac{d\psi_E}{dx} = -r_E \dot{m}_E''$$

where $\dot{m}_I''$ and $\dot{m}_E''$ are the mass transfer rates across inner and outer boundaries.

The effective viscosity is given by

$$\mu_{eff} = (\mu + \mu_t) \text{ and}$$

$$\epsilon = \mu_{eff}/\rho$$

The above governing equations can all be written in the general form given below

$$\frac{\partial \phi}{\partial x} + (a + b\omega) \frac{\partial \phi}{\partial \omega} = \frac{\partial}{\partial \omega} \left(c \frac{\partial \phi}{\partial \omega} \right) + d \quad (3.8)$$

where ϕ is any dependent variable.

The different terms in the above general mathematical expression pertaining to the various governing equations are listed below.

1. Momentum equation

$$a \equiv \frac{r_I \dot{m}_I''}{(\psi_E - \psi_I)}$$

$$b \equiv \frac{r_E \dot{m}_E'' - r_I \dot{m}_I''}{(\psi_E - \psi_I)}$$

$$c \equiv \frac{r^2 \mu_{eff} \rho u}{(\psi_E - \psi_I)^2}$$

$$d \equiv -\frac{1}{\rho u} \frac{dp}{dx}$$

2. Total enthalpy equation

$$a \equiv \frac{r_I \dot{m}_I''}{(\psi_E - \psi_I)}$$

$$b \equiv \frac{r_E \dot{m}_E'' - r_I \dot{m}_I''}{(\psi_E - \psi_I)}$$

$$c \equiv \frac{r^2 \mu_{eff} \rho u}{(\psi_E - \psi_I)^2 Pr_{eff}}$$

$$d \equiv \frac{\partial}{\partial \omega} \left\{ \frac{r^2 \rho u \mu_{eff}}{(\psi_E - \psi_I)^2} \left(1 - \frac{1}{Pr_{eff}} \right) \frac{\partial}{\partial \omega} \left(\frac{u^2}{2} \right) \right\}$$

3. TKE equation

$$a \equiv \frac{r \dot{m}_I''}{(\psi_E - \psi_I)}$$

$$b \equiv \frac{r \dot{m}_E'' - r \dot{m}_I''}{(\psi_E - \psi_I)}$$

$$c \equiv \frac{(\rho u)^2 \epsilon u}{Pr_K (\psi_E - \psi_I)^2}$$

$$d \equiv \frac{\tau r}{(\psi_E - \psi_I)} \frac{\partial u}{\partial \omega} + \frac{a_2 K^{3/2}}{1_K u}$$

Initial Profiles and Boundary Conditions

To solve the parabolic boundary layer equations, it is necessary to know the values of all the variables at a station upstream of the region of interest. These initial profiles are obtained from experiments, empirical relations, approximate theories or guess work. It is, however, desirable that the initial profiles should correspond to the real flow in certain gross features, for example, the mass, momentum or enthalpy fluxes implicit in the initial profiles should be approximately equal to those of the real flow under consideration.

Boundary Conditions

The flow regime of interest will be bounded by two imaginary surfaces I and E, the internal and external boundary. Three types of boundaries are of practical importance.

1. The boundary surface may coincide with a solid wall, resulting in conventional boundary layers over solid surfaces.
2. The boundary surface may be adjacent to an inviscid stream called a free boundary.
3. The boundary surface may coincide with a line of symmetry.

To solve the boundary layer equations we need in addition to the initial profiles, information relating to the fluxes across the I and E boundaries. This information is usually given in terms of the values of the dependent variables or their gradients along the boundary. In the particular case of a symmetry line boundary, the gradients normal to the line of symmetry are zero. In the case of a stationary wall boundary, the velocity along the wall is always zero, however it is often necessary to specify an effective slip velocity at the wall which corresponds to the flow outside the laminar boundary layer.

Because of the presence of pressure gradient term in the momentum conservation equation, the pressure variation with the downstream distance has to be prescribed. In the case of internal flow, the pressure is obtained in the course of the solution procedure since the cross-sectional area distribution of the duct is given.

Auxiliary Relationships

The dependent variables and fluid properties are linked by thermodynamic relationships. Neglecting the kinetic energy associated

with motion in y direction and also that of the turbulent fluctuations, the stagnation enthalpy is given by

$$\tilde{h} = h + \frac{u^2}{2}$$

Quantities like shear stress, diffusional flux and heat flux appear in the conservation equations. For laminar boundary layers, these quantities could have been expressed through Newton's Law of Viscosity, Fick's Law of Diffusion or Fourier's Law of Heat Conduction. For turbulent flows, it is convenient to assume effective values which also obey laws of similar form; the laminar exchange property is replaced by an effective exchange coefficient:

Thus, we have

$$\tau = \mu_{\text{eff}} \left(\frac{\partial u}{\partial y} \right) \text{ and} \quad (3.9)$$

$$J_h = - \left(\frac{\mu_{\text{eff}}}{Pr_{\text{eff}}} \right) \frac{\partial h}{\partial y} \quad (3.10)$$

where Pr_{eff} is the effective Prandtl number, which is approximately equal to 0.8 for pipe flow.

The expression for effective Prandtl number (Pr_{eff}) near the wall can be obtained from the following relationship (25).

$$\frac{\mu_{\text{eff}}}{Pr_{\text{eff}}} = \frac{\mu}{Pr} + \frac{1}{Pr_t} (\mu_{\text{eff}} - \mu) \quad (3.11)$$

where Pr and Pr_t are the laminar and turbulent Prandtl numbers respectively. For turbulent flows the reported value for Pr_t is about 0.8.

Theory of Law-Of-The-Wall

Generally it is possible to solve the governing equations once the initial profiles and boundary conditions are specified. But there is a difficulty associated with the wall region. Both the dependent variables and the effective exchange coefficients vary steeply in this region, and as a consequence, no matter which numerical method we choose, good accuracy in this region can be assured only by devoting substantial computational effort to this narrow region. The calculation procedure overcomes this difficulty by the assumption that x-wise convection is negligible and thus, near the wall there exists a "Couette flow," i.e., a one-dimensional flow, in which the conditions are determined primarily by the fluxes of mass, momentum and energy across the layer. Here the mathematical problem is to solve ordinary rather than partial differential equations. The Couette flow solutions can be carried out once and for all and the solutions can be used as asymptotes or boundary conditions for the partial differential equations. Couette flow solutions can be expressed ultimately in the form of algebraic relationships.

Under the limiting case of Couette flow, the boundary layer equations reduce to the following form (25).

$$\rho v \frac{\partial u}{\partial y} = - \frac{dp}{dx} + \frac{d\tau}{dy} \quad (3.12)$$

$$\rho v \frac{\partial \tilde{h}}{\partial y} = - \frac{d}{dy} (J_h - u\tau) \quad (3.13)$$

where it is assumed that the radius of curvature is negligible. The above equations can be readily integrated to yield

$$\tau(y) = \tau_w + \dot{m}_w'' u(y) + \frac{dP}{dx} y \text{ and} \quad (3.14)$$

$$J_h(y) = J_{h,w} + \dot{m}_w'' (\tilde{h}_w - \tilde{h}(y)) + u(y) \tau(y) \quad (3.15)$$

where $\dot{m}_w'' = (\rho v)$ at the wall

Patankar and Spalding (25) cast these equations in the non-dimensional form, introducing dimensionless groups, as follows.

$$\tau^+ = 1 + p^+ y^+ + m^+ u^+ \quad (3.16)$$

$$J^+ = 1 + m^+ \phi^+ - W u^+ \tau^+ \quad (3.17)$$

where

$$\tau^+ = \frac{\tau(y)}{\tau_w}$$

$$y^+ = y (\tau_w \rho)^{1/2} / \mu_c$$

$$u^+ = u / (\tau_w / \rho)^{1/2}$$

$$p^+ = \mu \frac{dP}{dx} / (\rho \tau_w^3)^{1/2}$$

$$J^+ = J_h / J_{h,w}$$

$$m^+ = \dot{m}_w'' / (\tau_w \rho)^{1/2}$$

$$W = (\tau_w^3 / \rho_c)^{1/2} / J_{h,w}$$

$$\mu^+ = \mu_{eff} / \mu_c$$

$$\phi^+ = (\tilde{h}_w - h) (\tau_w \rho_c)^{1/2} / J_{h,w}$$

From the above non-dimensionalization we get a straight forward relation for the shear stress, viz:

$$\tau^+ = \mu^+ \frac{du^+}{dy^+} \quad (3.18)$$

Under the Couette flow assumptions, we have the relation

$$\tau(y) = \tau_w + \mu_w'' u(y) + \frac{dp}{dx} y$$

which shows that the shear stress is a linear function of y . For no mass transfer across the walls, we have

$$\frac{dp}{dx} = \frac{\tau(y) - \tau_w}{y} \quad (3.19)$$

Assuming that $\tau(y) = \tau_w f(y)$, then

$$\frac{dp}{dx} = \tau_w \left(\frac{f(y) - 1}{y} \right) \quad (3.20)$$

Since $\frac{dp}{dx}$ is a function of x only, the right hand side of the equation has to be a constant, that

$$\frac{dp}{dx} = \tau_w \left(\frac{f(y) - 1}{y} \right) = \text{Constant} \quad (3.21)$$

or $f(y) = 1 + cy$. It follows from this relation that

$$\tau(y) = \tau_w (1 + cy) \quad (3.22)$$

It can also be written that

$$\tau^+ = 1 + c^+ y^+ \quad (3.23)$$

where $c^+ = c/(\tau_w \rho)^{1/2}/\mu$

The basic equation for the Couette flow with no mass transfer can be written as

$$\frac{d\tau}{dy} = \frac{dP}{dx}$$

$$\frac{d\tau}{dy} = c\tau_w = \frac{dP}{dx} \quad \text{or}$$

$$c = \frac{dP}{dx} / \tau_w$$

It can be noted that the shear stress distribution is a linear function of the pressure gradient. The viscosity distribution can easily be found from the Couette flow analysis because

$$\mu^+ = \tau^+ / \frac{\partial u^+}{\partial y^+} = \frac{1 + p^+ y^+}{\frac{du^+}{dy^+}} \quad (3.24)$$

At this point the velocity distribution is not known. We make the assumption that in the Couette regime, the velocity profile is the same as a turbulent boundary layer.

Thus

$$u^+ = y^+ \quad \text{for } y^+ < 10$$

$$u^+ = \frac{1}{k} \log_e (y^+) + B \quad \text{for } y^+ > 10 \quad (3.25)$$

Hence

$$\frac{du^+}{dy^+} = 1 \quad \text{for } y^+ < 10$$

and

$$\frac{du^+}{dy^+} = \frac{1}{ky^+} \quad \text{for } y^+ > 10 \quad (3.26)$$

Then the viscosity distribution can be written as

$$\mu^+ = \tau^+ = 1 + p^+ y^+ \quad \text{for } y^+ \leq 10 \quad (3.27)$$

$$\text{and } \mu^+ = \frac{\tau^+}{\left(\frac{du^+}{dy^+}\right)} = ky^+ (1 + p^+ y^+) \quad \text{for } y^+ > 10 \quad (3.28)$$

Thus, we can plot the μ^+ distribution for the logarithmic layer for a Couette flow having adverse pressure gradient. Figure 9 gives the viscosity variation in the sublayer and Figure 10 gives the same for the logarithmic layer. The plots show that the effective viscosity can become several orders of magnitude greater than the laminar viscosity for a retarded Couette flow.

It can be shown that the effect of pressure gradient on viscosity is very large for large positive pressure gradients.

Suppose $P^+ \approx 1$ (a severe condition as Figures 8 and 9 suggest)

$$\text{then, } \mu \frac{dP}{dx} / (\rho \tau_w^3)^{1/2} \approx 1 \quad (3.29)$$

$$\text{or } \frac{dP}{dx} = (\rho \tau_w^3)^{1/2} / \mu$$

$$= \frac{\tau_w u_\tau}{\nu}$$

$$\text{Since } \tau_w = C_f \frac{1}{2} \rho u^2$$

$$C_f = 2 \frac{u_\tau^2}{u^2}$$

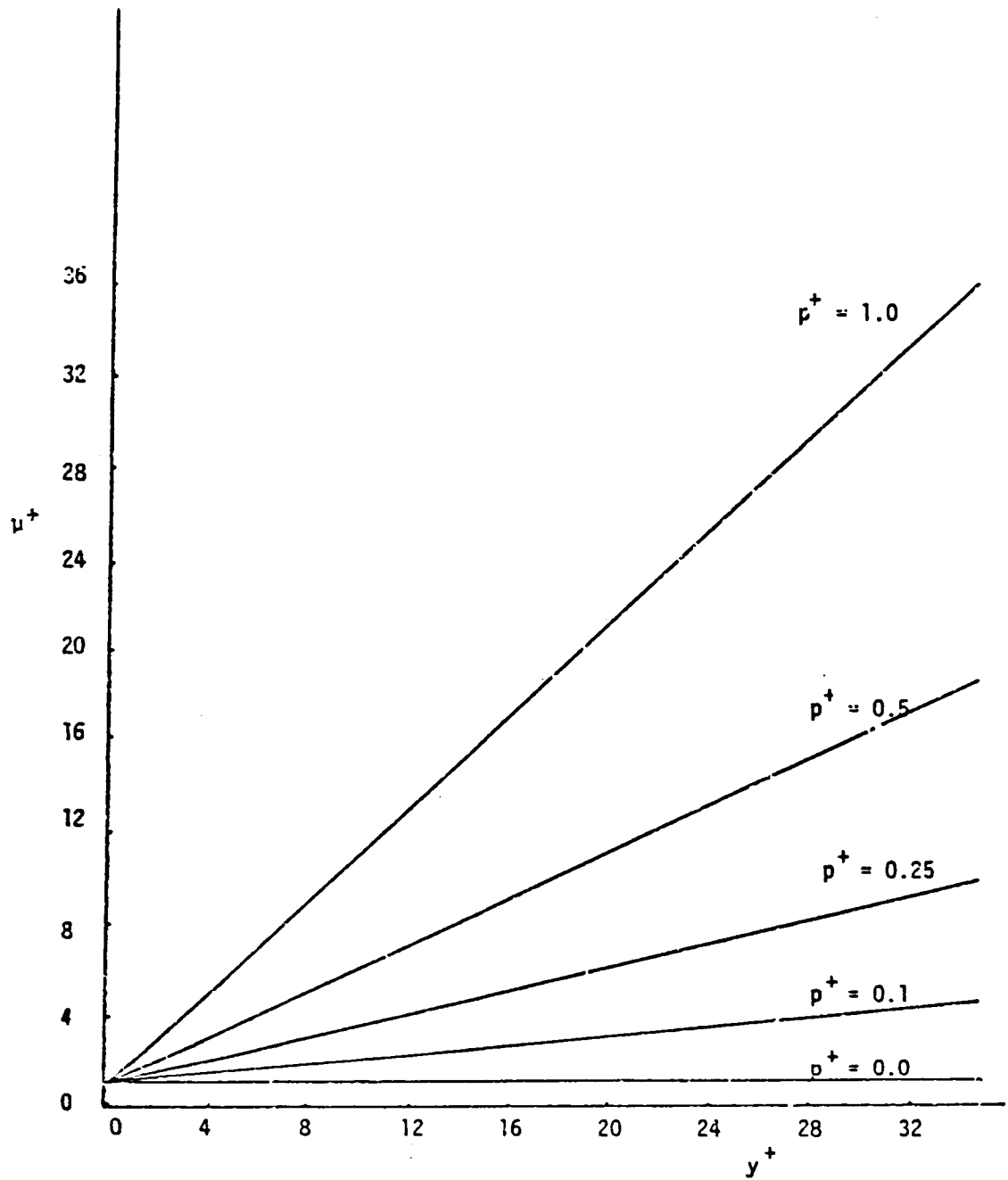


Figure 9. Viscosity Variation in Laminar Sublayer for Positive $\frac{dp}{dx}$

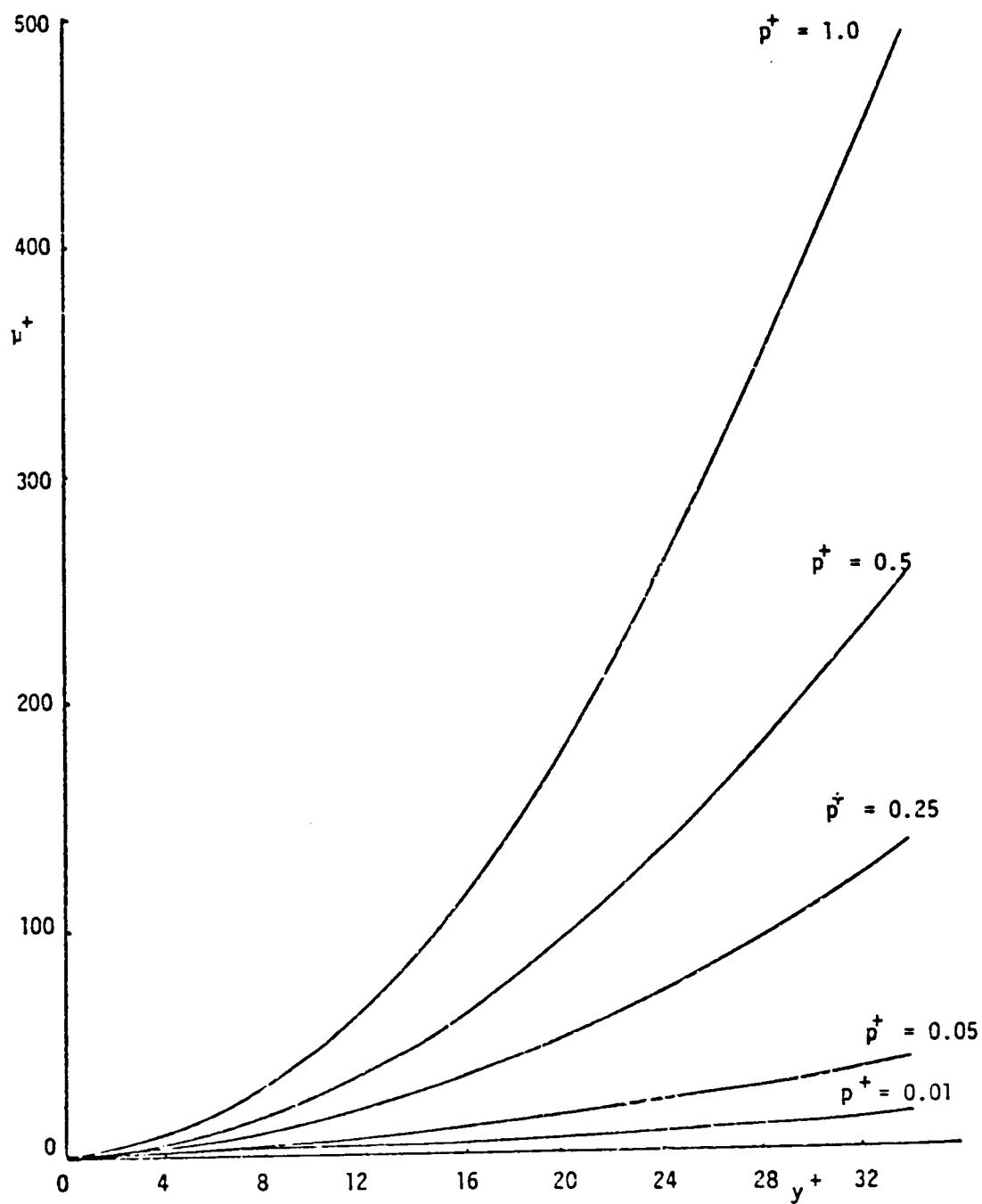


Figure 10. Viscosity Variation in Log Layer for Positive $\frac{dp}{dx}$

$$\tau_w = 2 \frac{u_\tau^2}{u^2} \cdot \frac{1}{2} \rho u^2 = \rho u_\tau$$

$$\text{hence } \frac{dP}{dx} \approx \rho u_\tau^3 / \nu \quad (3.30)$$

This is a condition signifying that pressure gradient effects are being felt by the boundary layer.

Treatment Of The Wall Region

Before discussing the wall function relationships, that enter the Couette flow model of the wall, it is necessary to describe the grid scheme for the finite difference computation. The scheme of the subscripts to identify the ω distribution is given in Figure 11. The interval between $\omega=0$ and $\omega=1$ is divided into N strips. Subscripts 1 and 2 denote the true and ship value respectively at the I boundary and the subscript 2.5 refers to a line midway between the I boundary and grid line 3; similarly $N + 1.5$ is used for a line midway between $N + 1$ and E boundary. Lines with subscripts 2.5 and $N + 1.5$ have special significance because they represent the boundaries between which lie all the control volumes used for the difference equations. ϕ_2 is defined as the slip value for the I boundary which enables us to obtain from a linear ϕ - ω interpolation between points 2 and 3, the correct slope and the value of ϕ at the point 2.5.

A brief outline of the wall function relationships used in the analysis is given below (Reference 25).

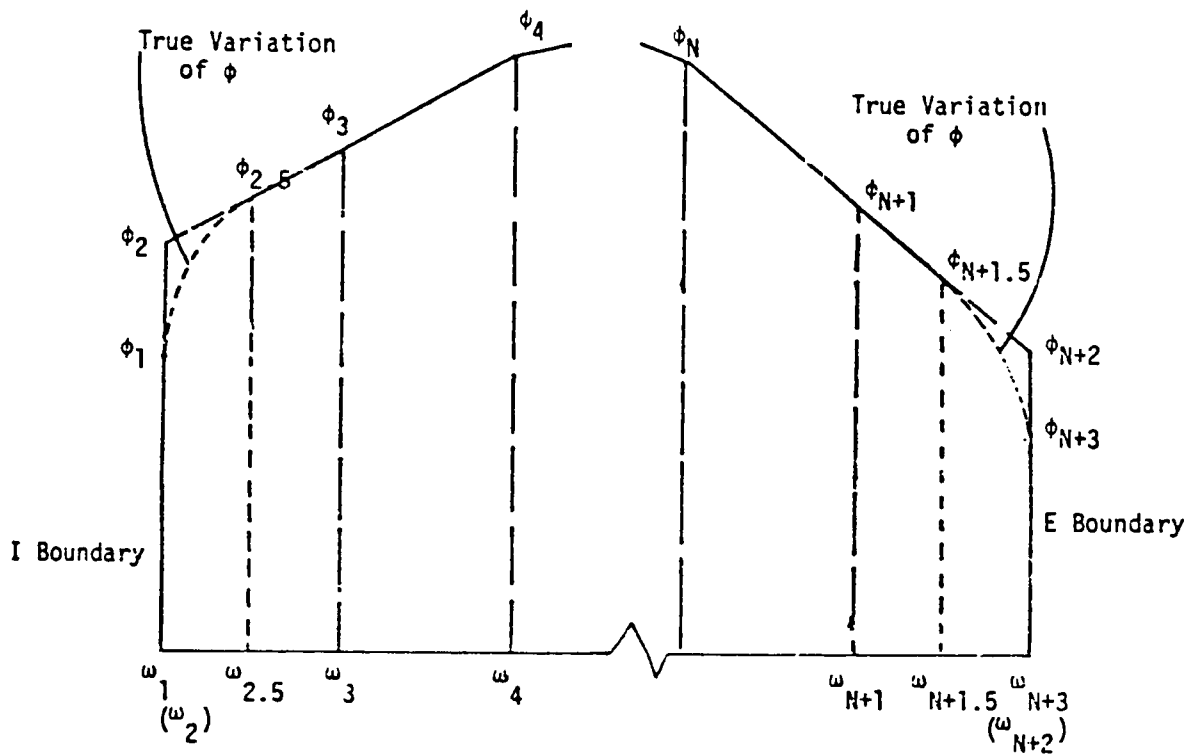


Figure 11. Scheme of Subscripts for the True and Slip Values of ϕ

Define a drag coefficient as

$$s = \tau_w / \rho_c u_c^2 \quad (3.31)$$

where subscript c denotes the Couette flow conditions.

The momentum flux across a shear boundary layer is given by

$$\tau_I r_I = s (\rho u^2 r)_{2.5} \quad (3.32)$$

For a wall boundary it can be written that

$$\tau_w r_w = s (\rho_R u_R^2 r_R) \quad (3.33)$$

where the subscript R denotes any reference condition in the turbulent law-of-the-wall regime in the boundary layer. The above expression can be written as

$$s^{1/2} = \frac{u_I}{u_R} \left(\frac{r_w}{r_R} \right)^{1/2} \quad (3.34)$$

The classical law-of-the-wall expression for a turbulent boundary layer is transformed to the following form

$$u^+ = \frac{1}{k} \log_e (E y^+) \quad (3.35)$$

where E is the transformed additive constant B is the log law

$$u^+ = \frac{1}{k} \log_e (y^+) + B$$

Introducing the drag coefficient s, equation 3.35 was written by Patankar and Spalding (25) as

$$s^{1/2} = k / \log_e (E R_R s^{1/2}) \quad (3.36)$$

where R_R is a reference Reynold's number.

The above equation is solved iteratively for s and the friction velocity and wall shear stress are determined from the relationships.

$$u_\tau = s^{1/2} u_R \left(\frac{r_R}{r_w} \right)^{1/2} \text{ and}$$

$$\tau_w = \rho_R s u_R^2 \left(\frac{r_R}{r_w} \right) \quad (3.37)$$

When the mass transfer parameter M defined as $M \equiv m_I''/(\rho u)_{2.5}$ is not zero, Spalding and Patankar evaluated the drag coefficient from a more complex expression

$$s = \left[\frac{k}{\log_e(ER_R s^{1/2})} \left\{ 1 - \frac{\ln}{4} \left\{ \frac{\log_e(ER_R s^{1/2})}{k} \right\}^2 \right\}^2 \right] \quad (3.38)$$

For laminar flow and uniform property boundary layer, the drag coefficient is expressed by

$$s = \frac{M[1 - (e^{MR_R} - 1 - MR_R) F/(M^2 R_R)]}{(e^{MR_R} - 1)}$$

$$\text{where } F \equiv \left[\left(\frac{dp}{dx} \right) (\rho/\rho u^2) \right]^{2.5} \quad (3.39)$$

The wall function relationships for other dependent variables is given below.

For stagnation enthalpy, we can write the diffusional flux through I boundary as

$$J_{\text{diff},h,I} = g_{h,I} \{ \tilde{h}_1 - \tilde{h}_{2.5} + (1 - H) u_{2.5}^2/2 \} \quad (3.40)$$

where H is the recovery factor for Couette flow region and

$$g_{h,I} = S_t (\rho u)^{2.5}$$

where S_t = Stanton number

The Stanton number for a turbulent uniform property Couette flow can be expressed as

$$S_t = \frac{s}{Pr_t (1 - s^{1/2} P)} \quad (3.41)$$

where Pr_t = turbulent Prandtl number

and P is a factor primarily dependent on pressure gradient

The boundary condition for the turbulent kinetic energy is that TKE is always zero at the walls, although at the wall matching point it will be significantly greater than zero.

Modifications on TKE Dissipation Length Scale Distribution

Before discussing the improvements on the dissipation length scale, it is necessary to establish the relationship between Prandtl Mixing Length and the TKE Dissipation Length Scale.

An algebraic relationship is derived here to relate the magnitudes of the Prandtl Mixing Length and the length scale in the dissipation term of the turbulent kinetic energy equation. The technique for derivation is to use the relations for turbulent shear stress under the assumption that the turbulent boundary layer is near equilibrium.

We can write the following relationships for shear stress

$$1. \quad \tau_t = \rho l_p^2 \left| \frac{\partial u}{\partial y} \right| \frac{\partial u}{\partial y}$$

Prandtl Mixing Length

$$2. \quad \tau_t = \rho a_1 K$$

Bradshaw

When equilibrium conditions prevail, we have turbulent production = kinetic energy dissipation.

or

$$\tau_t \frac{\partial u}{\partial y} = \frac{\rho a_2 K^{3/2}}{l_k} \quad (3.42)$$

Solving for l_k , we have

$$l_k = \rho a_2 K^{3/2} / \rho a_1 K \frac{\partial u}{\partial y}$$

Substituting for τ_t from the Bradshaw hypothesis

$$l_k = \rho a_2 K^{3/2} / \rho a_1 K \frac{\partial u}{\partial y}$$

$$l_k = \frac{a_2}{a_1} K^{3/2} / \frac{\partial u}{\partial y}$$

which can be written as

$$l_k = \frac{a_2}{a_1} \frac{(a_1)^{1/2} (K)^{1/2}}{\frac{\partial u}{\partial y}}$$

Equating 1 and 2 and rearranging, we get

$$l_p = \frac{(a_1)^{1/2} (K)^{1/2}}{\frac{\partial u}{\partial y}}$$

The relationship between l_k and l_p , is thus obtained as given by

$$l_k = \left(\frac{a_2}{a_1 (a_1)^{1/2}} \right) l_p \quad (3.43)$$

If we assume that both length scales are equal, i.e., $l_p = l_k$

$$\frac{a_2}{a_1 (a_1)^{1/2}} = 1 \quad (3.44)$$

The value of a_1 has been found experimentally to be approximately 0.3 over a significant number of boundary layers and free jets. Hence, the value of a_2 is thus

$$a_2 = 0.164$$

A value of $a_2 = 0.169$ (scaled to 1.69) was used in the original calculation procedure, based on reference (1) in which the model was developed from free shear layer analysis.

Mixing Length Distribution in the Wall Region

In the previous section, it has been stated that the Prandtl Mixing Length and the length scale for the dissipation of turbulent kinetic energy are directly related. In this section we examine one of the proposals for modifying the mixing length distribution in the wall region of turbulent boundary layers.

It has been observed that, in severe adverse pressure gradients, universal law-of-the-wall breaks down as shown in Figure 5. The breakdown from the universal law-of-the-wall can either be a general breakdown or a progressive breakdown. In either case, departures from the log law will become noticeable progressively closer to the wall with increasingly adverse pressure gradient.

Based on experimental evidence Mc D Galbraith, et al.(12) have suggested that simple expression $l=ky$ for the mixing length in the wall region is inadequate and its validity is restricted to where the shear stress is substantially constant through the wall region. In practical cases the shear stress varies substantially in the wall region and according to Reference 12, the relation $l=ky$ must be replaced by $l=ky (\tau/\tau_w)^{1/2}$. A closed form expression for τ/τ_w was given in Reference 12 as

$$\frac{\tau}{\tau_w} = 1 + p^+ y^+ + \frac{1}{2} a^+ [c_1 + y^+ (c_2 (\log_e y^+)^2 + c_3 \log_e y^+ + c_4)] \quad (3.45)$$

$$\text{where } y^+ = \frac{u_\tau y}{\nu}$$

$$p^+ = \frac{\nu}{\rho u_\tau^3} \frac{dp}{dx}$$

$$a^+ = \frac{\nu}{\rho u_\tau^3} \frac{d\tau_w}{dx}$$

$$\tau_w = \rho u_\tau^2$$

The values of the constants were $C_1 = -777.0$; $C_2 = 5.706$; $C_3 = 14.62$, and $C_4 = 15.08$.

It has been found in Reference 12 that (τ/τ_w) can be computed at $y/\delta = 0.03$ for the cases of boundary layers with adverse pressure gradients which they analyzed.

Reference (12) also suggests that universal law-of-the-wall has a wider range of validity than the hypothesis that $l=ky$ such that an effective value of k has to be computed from

$$k_{\text{eff}} = k (\tau/\tau_w)^{1/2}$$

In the present method of analysis k is treated as a universal constant, $k=0.41$.

However, in the previous section, we have established an equivalence between Prandtl Mixing Length and the dissipation length for turbulent kinetic energy. It has been assumed that the dissipation length scale also can be modified taking into account the effect of adverse pressure gradient. Thus, the dissipation length scale given in Reference (1) is modified as

$$l_k \text{ modified} = l_k (\tau/\tau_w)^{1/2} 0.035\delta \quad (3.46)$$

Numerical Solution Procedure

The general form of the governing equation is given by equation 3.8., i.e.,

$$\frac{\partial \phi}{\partial x} + (a + b\omega) \frac{\partial \phi}{\partial \omega} = \frac{\partial}{\partial \omega} \left(c \frac{\partial \phi}{\partial \omega} \right) + d$$

where x and ω are two independent variables or coordinates, a and b are arbitrary functions of the coordinate x and c and d may be arbitrary functions of any dependent and independent variables and ϕ stands for any one of a set of dependent variables, each possessing its own differential equation.

The general equation is parabolic and two-dimensional, enabling the use of "marching integration." Marching integration is that kind of integration which starts by determining the values at one end of the domain, then determines the values over a front displaced just a little from that end, and so gradually moves the integration front towards the other end of the domain until the required values have been determined everywhere. The direction of march is always that of the predominant direction of flow.

Coordinate System

The coordinate system used has a particular importance in the finite difference procedure. The most common practice is to employ a rectangular grid in x-y coordinates as shown in Figure 12. This is very uneconomical for a boundary layer problem, for boundary layers grow in thickness in the downstream direction. If the grid spacing is such that a certain number of grid points lie within the thickness of the layer at the upstream end, this distribution will prove to be too fine and hence wasteful at the downstream end.

Considerable economy is achieved by defining a non-dimensional stream function ω , as the cross stream variable. ω is always zero at one edge of the boundary layer and is unity at the other end.

The Finite Difference Equations

Finite difference equations are formed from the integrals of the differential equations over small regions associated with the individual nodes of the grid.

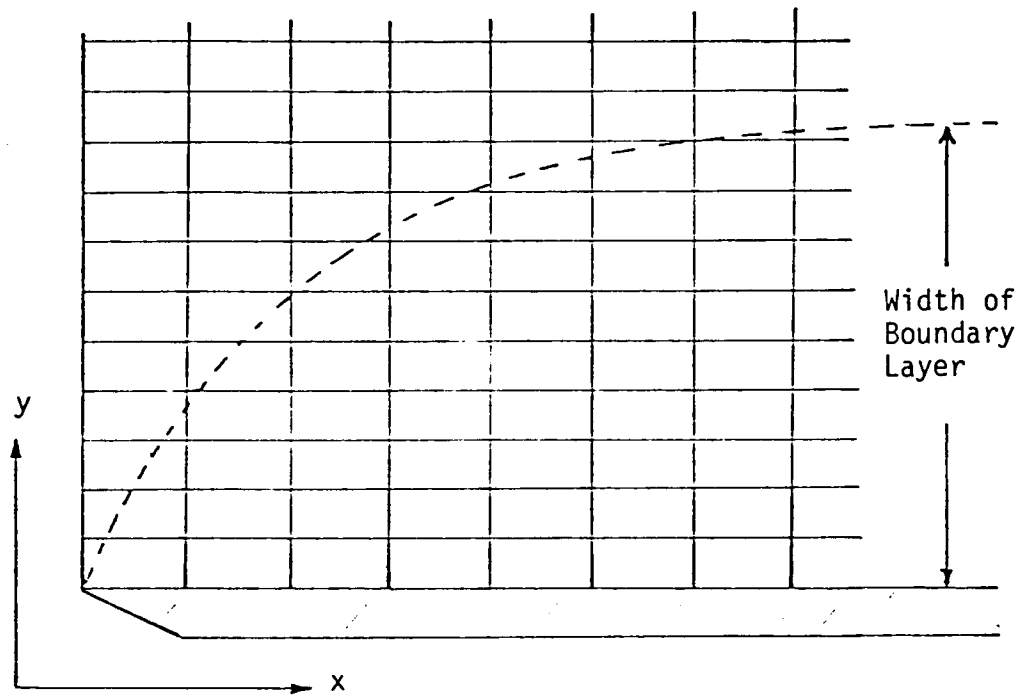


Figure 12. Illustration of the Unsuitability of a Cartesian Grid for a Growing Boundary Layer

The control volume and the relevant notations are given in Figure 13. Integration of the generation differential equation over the control volume gives

$$\int_{x_U}^{x_D} \int_{\omega_D-}^{\omega_D+} \frac{\partial \phi}{\partial x} d\omega dx / (x_D - x_U) (\omega_{D+} - \omega_{D-}) + \int_{x_U}^{x_D} \int_{\omega_D-}^{\omega_D+} (a + b\omega) \frac{\partial \phi}{\partial \omega} d\omega dx / (x_D - x_U) (\omega_{D+} - \omega_{D-}) = \int_{x_U}^{x_D} \int_{\omega_D-}^{\omega_D+} \frac{\partial}{\partial \omega} \left(c \frac{\partial \phi}{\partial \omega} \right) d\omega dx / (x_D - x_U) (\omega_{D+} - \omega_{D-}) + \int_{x_U}^{x_U} \int_{\omega_D-}^{\omega_D+} d \cdot d\omega dx / (x_D - x_U) (\omega_{D+} - \omega_{D-}) \quad (3.47)$$

Multiplying by $(\omega_{D+} - \omega_{D-})$ and simplifying, we get

$$\left[\int_{\omega_D-}^{\omega_D+} \phi_D \cdot d\omega - \int_{\omega_D-}^{\omega_D+} \phi_U d\omega \right] / \delta x + \left[\{(a + b\omega) \phi\}_{\omega_{D+}} - \{(a + b\omega) \phi\}_{\omega_{D-}} \right] - b \int_{\omega_D-}^{\omega_D+} \phi_D d\omega = \left\{ \left(c \frac{\partial \phi}{\partial \omega} \right)_{\omega_{D+}} - \left(c \frac{\partial \phi}{\partial \omega} \right)_{\omega_{D-}} \right\} + \int_{\omega_D-}^{\omega_D+} d \cdot d\omega \quad (3.48)$$

where $\delta x = (x_U - x_D)$

Multiplying by $(\Psi_E - \Psi_I)$ we get

$$\frac{(\Psi_E - \Psi_I)}{\delta x} \int_{\omega_D-}^{\omega_D+} \phi_D d\omega - \frac{\Psi_E - \Psi_I}{\delta x} \int_{\omega_D-}^{\omega_D+} \phi_U d\omega + (\Psi_E - \Psi_I) \left[\{(a + b\omega) \phi\}_{\omega_{D+}} - \{(a + b\omega) \phi\}_{\omega_{D-}} \right] - b(\Psi_E - \Psi_I) \int_{\omega_D-}^{\omega_D+} \phi_D d\omega = (\Psi_E - \Psi_I) \left\{ \left(c \frac{\partial \phi}{\partial \omega} \right)_{\omega_{D+}} - \left(c \frac{\partial \phi}{\partial \omega} \right)_{\omega_{D-}} \right\} + \int_{\omega_D-}^{\omega_D+} d \cdot d\omega \quad (3.49)$$

We introduce the following symbols

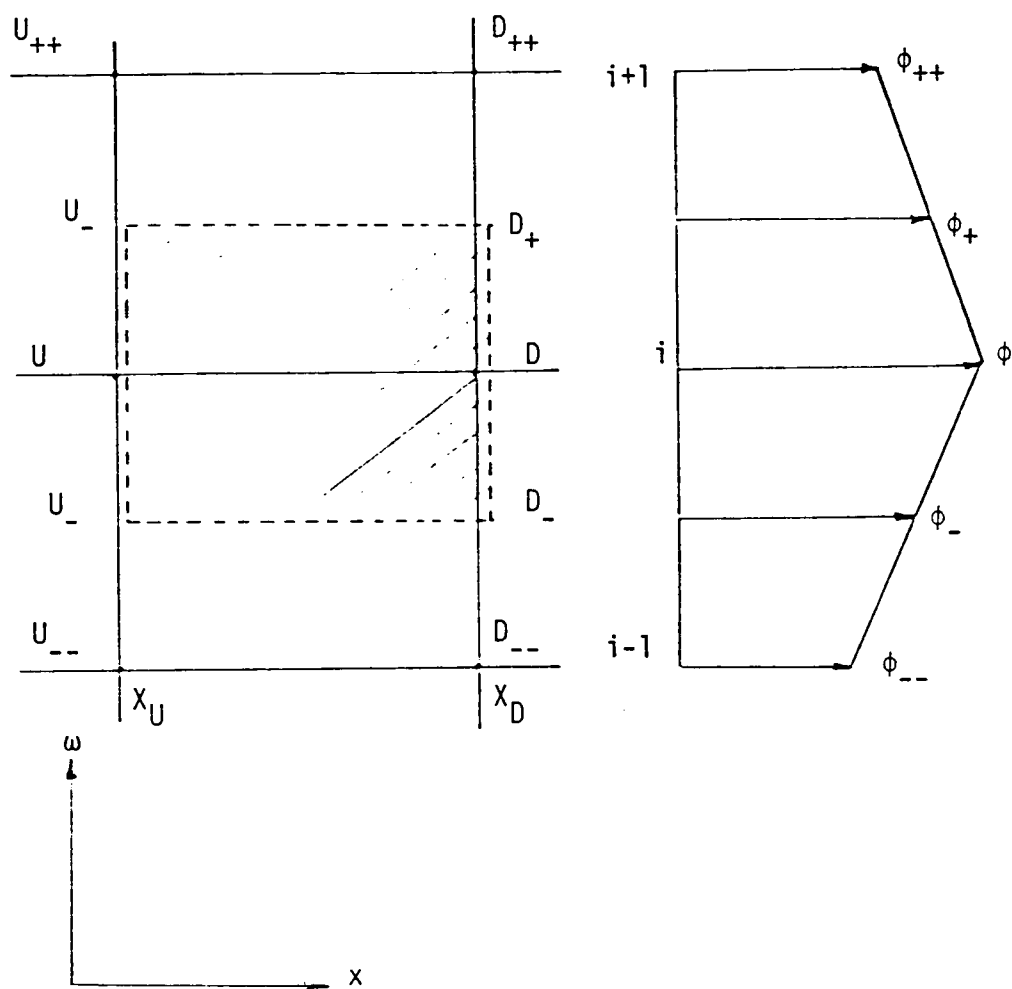


Figure 13. A Small Part of Grid Showing a Typical Control Volume

$$P \equiv (\Psi_E - \Psi_I) / \delta x$$

$$G \equiv (r_I \ddot{\eta}_I - r_E \ddot{\eta}_E) = - (\Psi_E - \Psi_I) b$$

$$L_+ \equiv (\Psi_E - \Psi_I) (a + b\omega_+)$$

$$= r_I \ddot{\eta}_I - \omega_+ + G$$

$$L_- \equiv r_I \ddot{\eta}_I - \omega_- - G$$

Equation 3.49 can be written as

$$\begin{aligned} (P+G) \int_{\omega_{D-}}^{\omega_{D+}} \phi_D d\omega - P \int_{\omega_{D-}}^{\omega_{D+}} \phi_U d\omega + L_+ \phi_+ - L_- \phi_- = \\ (\Psi_E - \Psi_I) \left\{ \left(c \frac{\partial \phi}{\partial \omega} \right)_{\omega_{D+}} - \left(c \frac{\partial \phi}{\partial \omega} \right)_{\omega_{D-}} \right\} + \int_{\omega_{D-}}^{\omega_{D+}} d \cdot d\omega \end{aligned} \quad (3.50)$$

Let

$$\Omega \equiv \omega_{++} - \omega_{--}$$

$$\Omega_+ \equiv \omega_{++} - \omega$$

$$\Omega_- \equiv \omega - \omega_{--}$$

Further, we have

$$\phi_{D-} = 1/2 (\phi_D + \phi_{D--})$$

$$\phi_{D+} = 1/2 (\phi_D + \phi_{D++})$$

Assuming that profiles are linear between the grid points and introducing the above symbols, the left hand side of the equation becomes

$$\begin{aligned} 2 \times \text{LHS} = \frac{1}{4} (P+G) (3\phi_D \Omega + \phi_{D++} \Omega_+ + \phi_{D--} \Omega_-) - \frac{1}{4} P \\ (3\phi_U \Omega + \phi_{U++} \Omega_+ + \phi_{U--} \Omega_-) + L_+ (\phi_D + \phi_{D++}) - L_- (\phi_D + \phi_{D--}) \end{aligned} \quad (3.51)$$

Rearranging, we get

$$2 \times \text{LHS} = \left\{ \frac{3}{4} (P+G)\Omega + L_+ - L_- \right\} \phi_D + \left\{ \frac{1}{4} (P+G)\Omega_+ + L_+ \right\} \phi_{D_{++}} \\ + \left\{ \frac{1}{4} (P+G)\Omega_{--} - L_- \right\} \phi_{D_{--}} - \frac{1}{4} P(3\phi_U\Omega + \phi_{U_{++}}\Omega_+ + \phi_{U_{--}}\Omega_-)$$

Now for the right hand side consider the diffusion term

$$(\Psi_E - \Psi_I) \left\{ \left(c \frac{\partial \phi}{\partial \omega} \right)_{\omega_{D_+}} - \left(c \frac{\partial \phi}{\partial \omega} \right)_{\omega_{D_-}} \right\}$$

Now, we know

$$(\Psi_E - \Psi_I) \left(c \frac{\partial \phi}{\partial \omega} \right)_{\omega_{D_+}} = (\Psi_E - \Psi_I) \left\{ \frac{r^2 \rho u \mu_{\text{eff}}}{(\Psi_E - \Psi_I)^2 \text{Pr}_{\text{eff}}} \frac{\phi_{D_{++}} - \phi_D}{\Omega_+} \right\} + \quad (3.52)$$

We know that $\frac{d\Psi}{dy} = \rho u r$

Hence $\frac{\Psi_{++} - \Psi}{y_{++} - y} = (\rho u r)_+$

But $\omega = \frac{\Psi - \Psi_I}{\Psi_E - \Psi_I} \rightarrow \Psi = \omega(\Psi_E - \Psi_I) + \Psi_I$

which gives

$$\Psi_{++} - \Psi = \omega_{++} (\Psi_E - \Psi_I) + \Psi_I - \omega(\Psi_E - \Psi_I) - \Psi_I = \Omega_+ (\Psi_E - \Psi_I)$$

Then

$$(\rho u r)_+ = \frac{\Omega_+ (\Psi_E - \Psi_I)}{(y_{++} - y)}$$

$$(\Psi_E - \Psi_I) \left(c \frac{\partial \phi}{\partial \omega} \right)_{\omega_{D_+}} = \left(\frac{r_+ \Gamma_{\text{eff}+}}{y_{++} - y} \right) (\phi_{D_{++}} - \phi_D) \quad (3.53)$$

where $\Gamma_{\text{eff}} = \left(\frac{\mu_{\text{eff}}}{Pr_{\text{eff}}}\right)$ Letting $T_+ = \left(\frac{r_+ \Gamma_{\text{eff}+}}{y_{++} - y}\right)$

$$(\Psi_E - \Psi_I) \left(c \frac{\partial \phi}{\partial \omega}\right)_{\omega_{D+}} = T_+ (\phi_{D++} - \phi_{D--}) \quad (3.54)$$

Similarly, we get

$$(\Psi_E - \Psi_I) \left(c \frac{\partial \phi}{\partial \omega}\right)_{\omega_{D-}} = T_- (\phi_D - \phi_{D--}) \quad (3.55)$$

where $T_- = \left(\frac{r_- - \Gamma_{\text{eff}-}}{y - y_{--}}\right)$

Now consider the source term $\int_{\omega_{D-}}^{\omega_{D+}} d \cdot d\omega$

Define $S \equiv (\Psi_E - \Psi_I) \int_{\omega_{D-}}^{\omega_{D+}} d \cdot d\omega$

From $\omega = \frac{\Psi - \Psi_I}{\Psi_E - \Psi_I}$, we have $\frac{d\Psi}{\Psi_E - \Psi_I} = \frac{\rho u dy}{(\Psi_E - \Psi_I)}$

Hence $S = \int_{\omega_{D-}}^{\omega_{D+}} \rho u r dy$.

The source term depends on which entity ϕ represents. For $\phi = u$, we have

$$S = \int_{\omega_{D-}}^{\omega_{D+}} \left(\frac{dp}{dx}\right) r dy \quad (3.56)$$

For stagnation enthalpy

$$S = \left\{ r \mu_{\text{eff}} \left(1 - \frac{1}{Pr_{\text{eff}}}\right) u \frac{\partial u}{\partial y} \right\}_+ - \left\{ r \mu_{\text{eff}} \left(1 - \frac{1}{Pr_{\text{eff}}}\right) u \frac{\partial u}{\partial y} \right\}_- \quad (3.57)$$

In each case S must be evaluated from a different formula.

But always we require a formula of the form

$$S = S_U + S_D \phi_D \quad (3.58)$$

Combining expressions 3.52, 3.54, 3.55, and 3.58, the finite difference equation can be written in the desired form

$$\phi = A\phi_{++} + B\phi_{--} + C \quad (3.59)$$

where $A = \frac{2T_+ - L_+ - 1/4(P+G)\Omega_+}{D}$

$$B = \frac{2T_- + L_- - 1/4(P+G)\Omega_-}{D}$$

$$C = \frac{1/4P(3\phi_1\Omega + \phi_{U+}\Omega_+ + \phi_{U-}\Omega_-) + 2S_U}{D}$$

$$D = 2(T_+ + T_-) + L_+ - L_- + 3/4(P+G)\Omega - 2S_D$$

Treatment of the Edge Region

Equation 3.59 allows ϕ values at all the grid points numbered 3 to $N+1$ inclusive, to be connected to the ϕ values of the neighboring points. The values for grid points 1 and $N+3$ are usually dictated by the boundary conditions. The formula to determine ϕ_2 and ϕ_{N+2} is discussed here.

The control volume and the relevant notations for ϕ_2 are given in Figure 14. The integral of the differential equation (3.8) from $\omega=0$ to $\omega=\omega_{2.5}$ after introducing the symbols, gives the finite difference equation

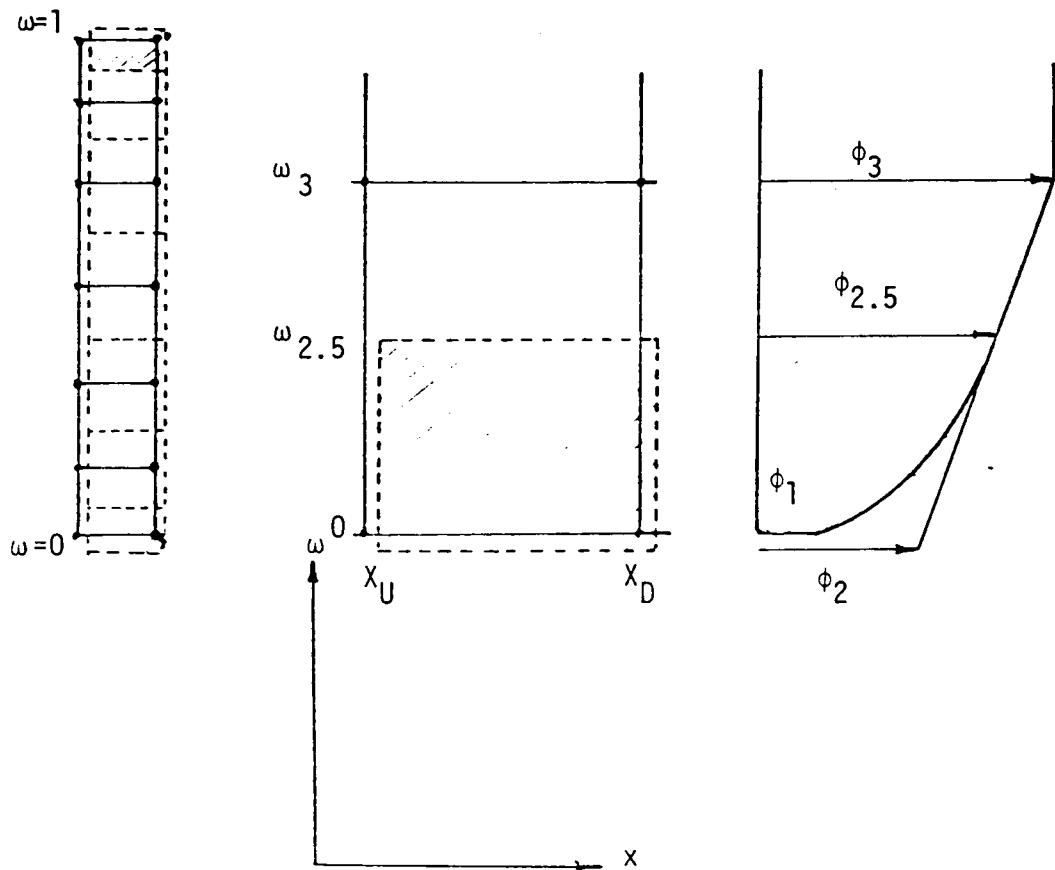


Figure 14. Illustration of Control Volume (Shaded)
Considered in the Derivation for ϕ_2

$$\phi_2 = A_2 \phi_3 + B_2 \phi_1 + C_2 \quad (3.60)$$

$$\text{where } A_2 = \frac{2T_1 - L_+ - T_1 - 1/4(P+G) \Omega_+ + 1/2 S_D}{D_2}$$

$$B_2 = \frac{2(T_1 + L_1)}{D_2}$$

$$C_2 = \frac{1/4 P(3\phi_{U2} + \phi_{U3}) \Omega_+ - 2T_1 \delta\phi_I + 2S_U}{D_2}$$

$$D_2 = 2T_+ + T_1 + L_+ + 3/4 (P+G) \Omega_+ - 3/2 S_D$$

Values of T_1 and $\delta\phi_I$ are determined from wall function relationships.

The corresponding finite difference equation for ϕ_{N+2} is

$$\phi_{NP2} = A_{NP2} \phi_{NP2} + B_{NP2} \phi_{NP3} + C_{NP2} \quad (3.61)$$

Solution of the Finite Difference Equations

We can express the algebraic equations resulting from the finite difference formulation as

$$\phi_i = A_i \phi_{i+1} + B_i \phi_{i-1} + C_i \quad (3.62)$$

where $i = 2, 2, 3 \dots N + 2$, the values of ϕ_1 and ϕ_{NP3} known from the boundary conditions. This system of equations can be arranged in the tridiagonal matrix form. The normal method of solution is matrix inversion technique. But in this system of equations only three unknowns appear in the particular order, in every equation. This allows the system of equations to be solved by successive substitution method. This is achieved as follows.

Equations 3.62 can be transformed into a simpler form

$$\phi_i = A_i' \phi_{i+1} + B_i' \quad (3.63)$$

where
$$A_i' = \frac{A_i}{1 - B_i A_{i-1}'}$$

$$B_i' = \frac{B_i B_{i-1}' + C_i}{1 - B_i A_{i-1}'}$$

$$A_2' = A_2$$

$$B_2' = B_2 \phi_1 + C_2$$

After calculation of A' and B' for $i = 2, 3, \dots, N+2$, ϕ can be obtained from equation 3.63 by successive substitution starting from NP3.

This computation procedure is a highly efficient one, both in time and storage.

Calculation of Cross Stream Distance

By definition, we have

$$\begin{aligned} \int_0^y \rho u dy &= (\psi - \psi_I) \\ &= (\psi_E - \psi_I) \omega \end{aligned} \quad (3.64)$$

We also have

$$r = r_I + y \cos \alpha \quad (3.65)$$

Combining these two relations, we get

$$y = 2I / \{r_I + (r_I^2 + 2I \cos \alpha)^{1/2}\} \quad (3.66)$$

where $I = (\Psi_E - \Psi_I) \int_0^{\omega} (\rho u)^{-1} d\omega$

for axisymmetric flow, $r_I = 0$

Hence,

$$y = \left(\frac{2I}{\cos \alpha} \right)^{1/2} \quad (3.67)$$

Treatment of Rectangular Geometry

The solution procedure as embodied in the computer code can solve only two-dimensional or axisymmetric boundary layer problems. But in many practical cases the diffuser cross section may not be circular. Hence, it is necessary to transform the actual geometry into a mathematically solvable geometry. This can be achieved by three methods.

Method I

In this method, the actual area of the diffuser section is transformed into an equivalent circular section so that the areas are equal (see Figure 15).

$$A = b \cdot h = \pi \text{ Req}^2$$

$$\text{Req} = \frac{b \cdot h}{\pi}$$

Req = Equivalent radius of a circular cross section

Method II

In this method the transformation is carried out as follows:

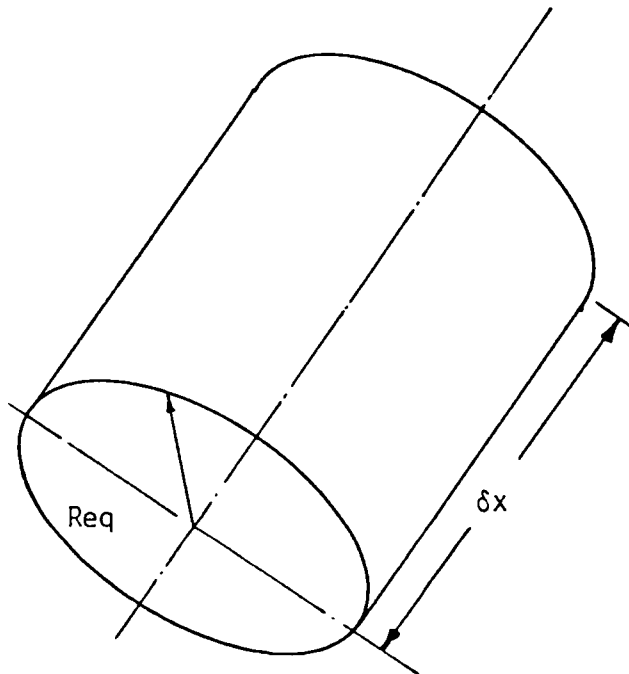
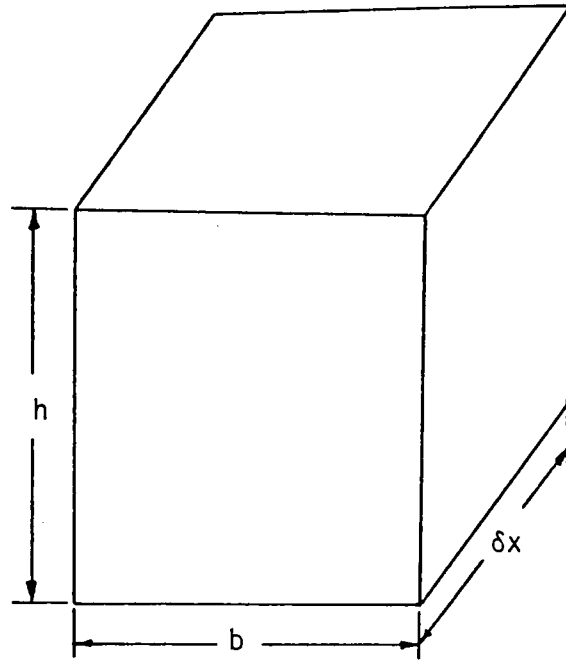


Figure 15. Schematic of Area Transformation

$$\delta A = 2h\delta x + 2b\delta x = (b + h)2\delta x$$

$$\delta A_{eq} = 2\pi R_{eq}\delta x$$

If $\frac{dA}{dx} \approx \frac{dA_{eq}}{dx}$ is taken as another criteria, we have

$$(b + h) 2 = 2\pi R_{eq}$$

$$R_{eq} = \left(\frac{b + h}{\pi}\right) \quad (3.69)$$

This is yet another way of defining an equivalent stream tube geometry. This method may be more accurate if frictional losses and heat transfer are predominant, and the stream wise pressure distribution is not needed with great accuracy.

$$A_{eq}(x) = \pi R_{eq}^2(x) = \left(\frac{b + h}{\pi}\right)^2$$

Method III

This method uses the classical hydraulic radius to replace the actual three-dimensional geometry. The definition for the hydraulic radius is

$$\begin{aligned} R_{eq}(x) &= \frac{\text{Cross-sectional Area}}{\text{Wetted Perimeter}} \\ &= \frac{b \cdot h}{2(b + h)} \end{aligned} \quad (3.70)$$

CHAPTER IV

RESULTS AND DISCUSSION

The modifications proposed for the Harsha-Glassman version of the diffuser code was implemented and the code was tested against available flow field data on axisymmetric diffusers. The results obtained are discussed.

The dissipation length scale was modified according to the procedure described in Chapter III. The correction factor $(\tau/\tau_w)^{1/2}$ computed based on the values computed at the previous integration step. The computation of slip velocity $u(NP2)$ is very important because it determines the slope at the velocity profile at the wall. For $u(NP2)$ that lies in the law-of-the-wall regime, $u(NP2)$ is computed from the relationship

$$u(NP2) = \frac{u_\tau}{k} \log_e \left(\frac{\Delta y u_\tau}{\nu} \right) + 5.5 u_\tau \quad (4.1)$$

where $\Delta y = y(NP3) - y(NP2)$

If $y(NP2)$ lies in the laminar regime $u(NP2)$ is computed from the relationship

$$u^+ = y^+$$

From which it follows that

$$u(NP2) = \frac{u_\tau^2 \Delta y}{\nu}$$

The value of u_t that is used in the above procedure is determined at each axial station of the flow by an iteration procedure. Velocity at the point midway between $y(NP1)$ is used to compute the reference Reynold's number, that is then used to evaluate whether the wall matching velocity distribution is to be linear (laminar regime) or logarithmic (fully turbulent). Then u_t is determined using the appropriate velocity distribution.

Results

Test cases were computed using the present method of analysis. Only limited experimental data is available on the flow field measurements in axisymmetric diffusers. Most of the investigations have concentrated on performance parameters, such as recovery and efficiency. However, Volume II of the Stanford Conference Proceedings on Computation of Turbulent Boundary Layers (31) contains data on axisymmetric diffusers listed as case 5000 and 5100. The tabulated data included mean velocity profiles, wall shear stress distributions, boundary layer mean velocity profiles and shape factor distributions. Case 5000 flow can be regarded as effectively incompressible with an inlet centerline Mach number = 0.149, which corresponds to $u = 170$ ft/sec.

To carry out computations on this axisymmetric diffuser, boundary layer thickness δ at $x = 3.6$ inches was taken as the input boundary layer thickness to the code. To generate the initial velocity profile from the boundary layer thickness δ , a power law was used. It was

found that 1/9th power best fits the data. The actual input profile and the power law profile are compared in Figure 16.

The initial TKE profile was obtained from the inlet velocity profile using Prandtl's Mixing Length Hypothesis. For example,

$$\tau_t = a_1 \rho K = \rho l_p^2 \left(\frac{\partial u}{\partial y} \right)^2 \quad (4.2)$$

The experimental and theoretical velocity profiles for Case 5000 in Reference 31 are compared in logarithmic coordinates in Figures 17 to 24. The agreement is quite satisfactory in the entire boundary layer at the downstream section of the diffuser, however, the agreement is less satisfactory in the wake region at the downstream section of the diffuser. It can be noted that even in this severe adverse pressure gradient flow there is a well defined logarithmic region.

The experimental and theoretical velocity profiles for Case 5000 in Reference 31 are compared in physical coordinates Figures 25 to 32. The agreement may be considered satisfactory in the near wall region, however, the agreement is less satisfactory in the wake region.

Even though the velocity distribution in logarithmic coordinates apparently gives a very close comparison, it was noted that the same velocity distribution in physical coordinates appears less satisfactory. This observation leads to the conclusion that for the

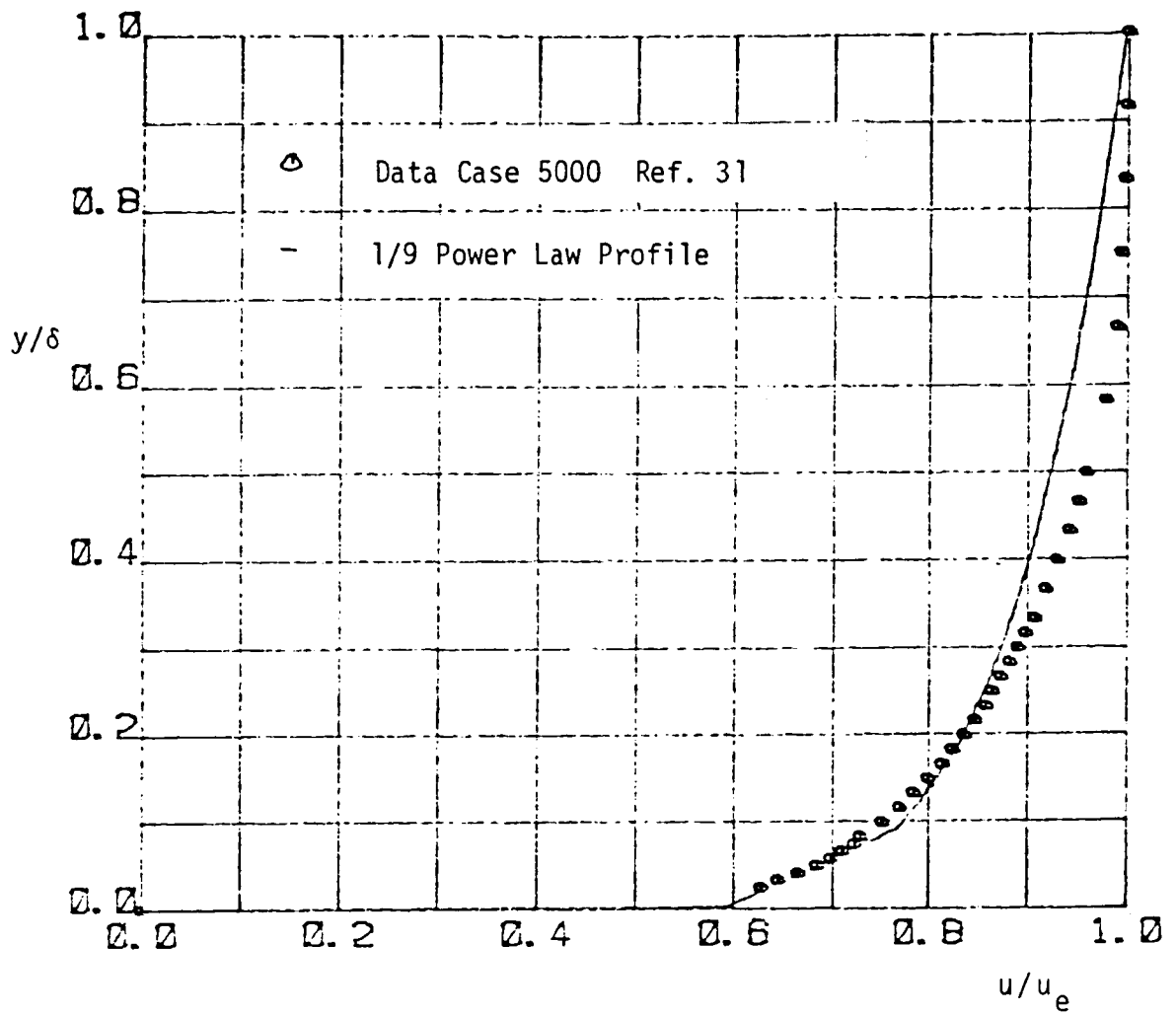


Figure 16. Comparison of Power Law Profile with Actual Profile

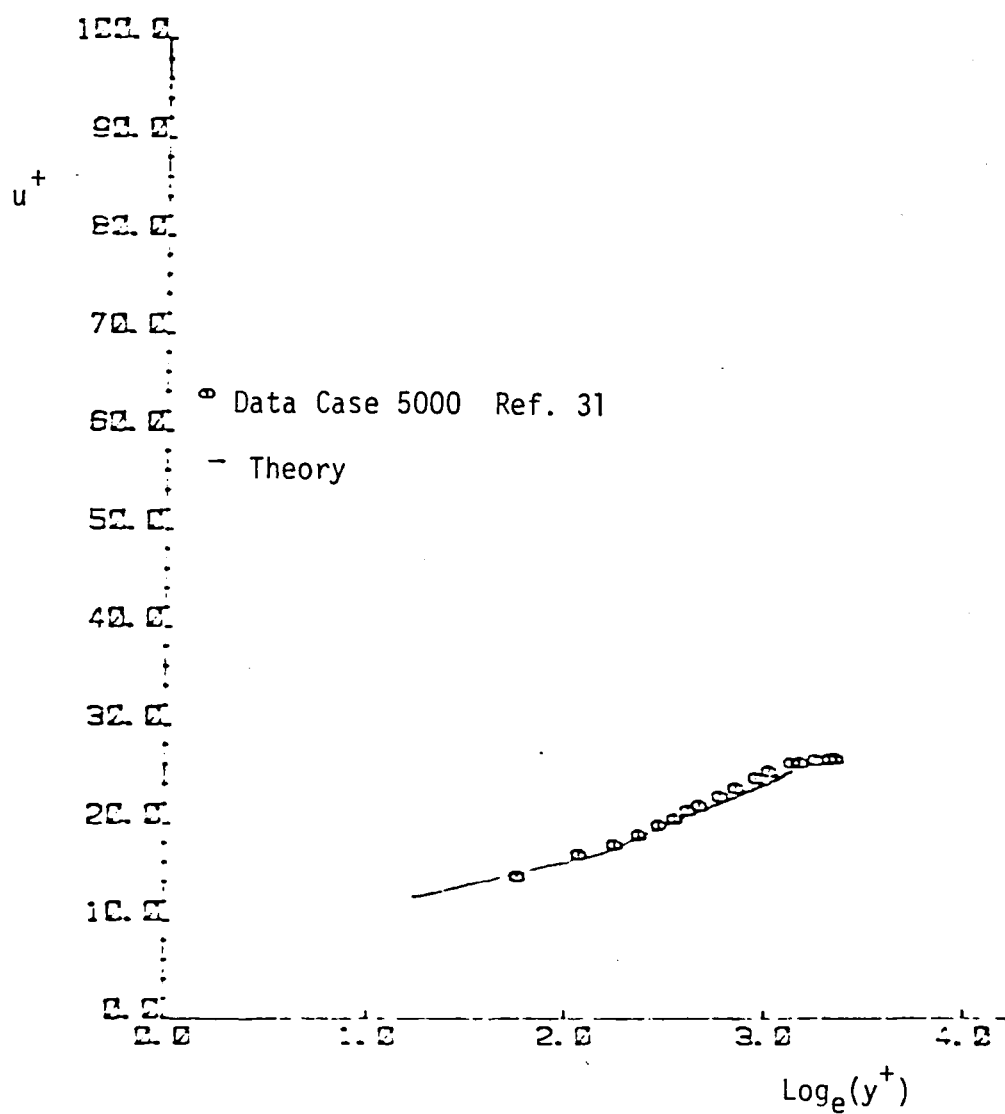


Figure 17. Comparison of Logarithmic Velocity Distribution at $x = 8.299$ Inches

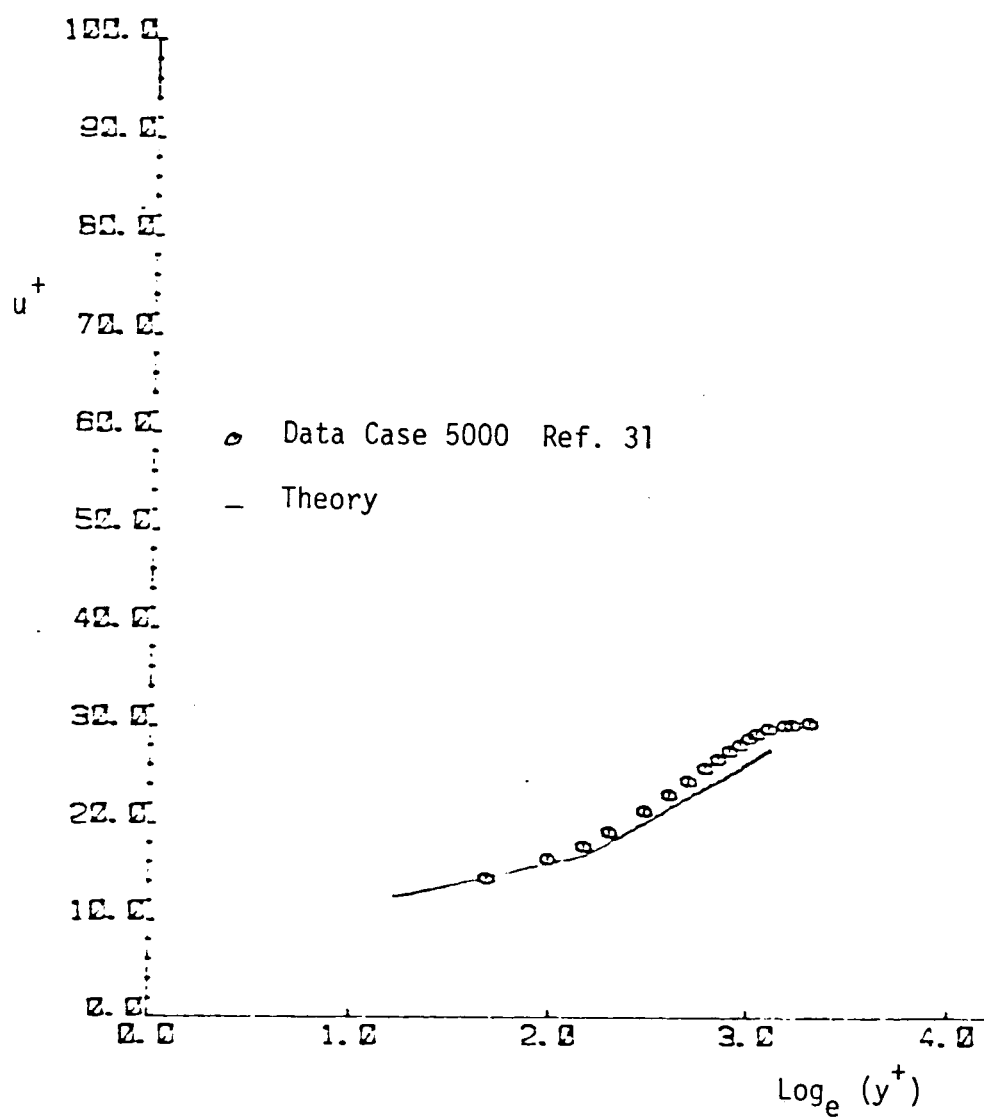


Figure 18. Comparison of Logarithmic Velocity Distribution at $x = 9.299$ Inches

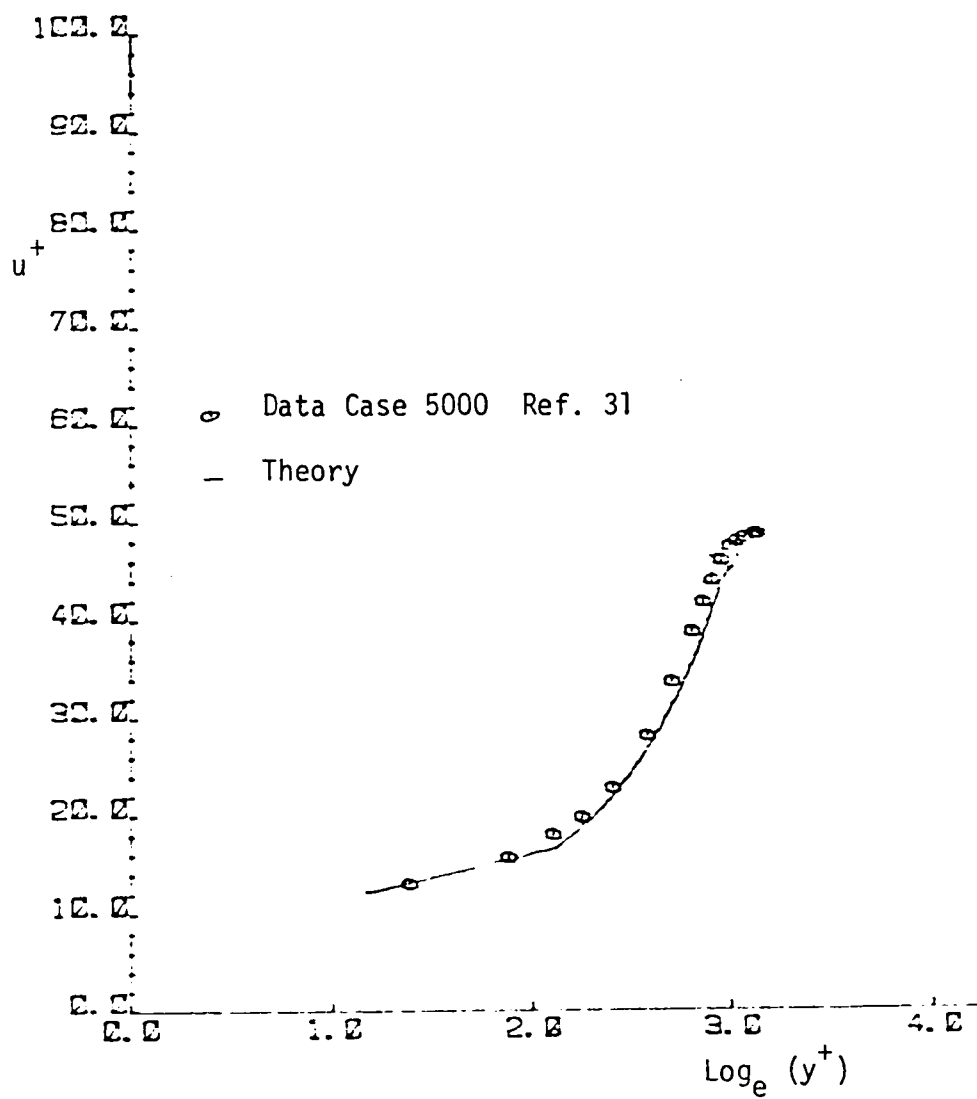


Figure 19. Comparison of Logarithmic Velocity Distribution at $x = 11.799$ Inches

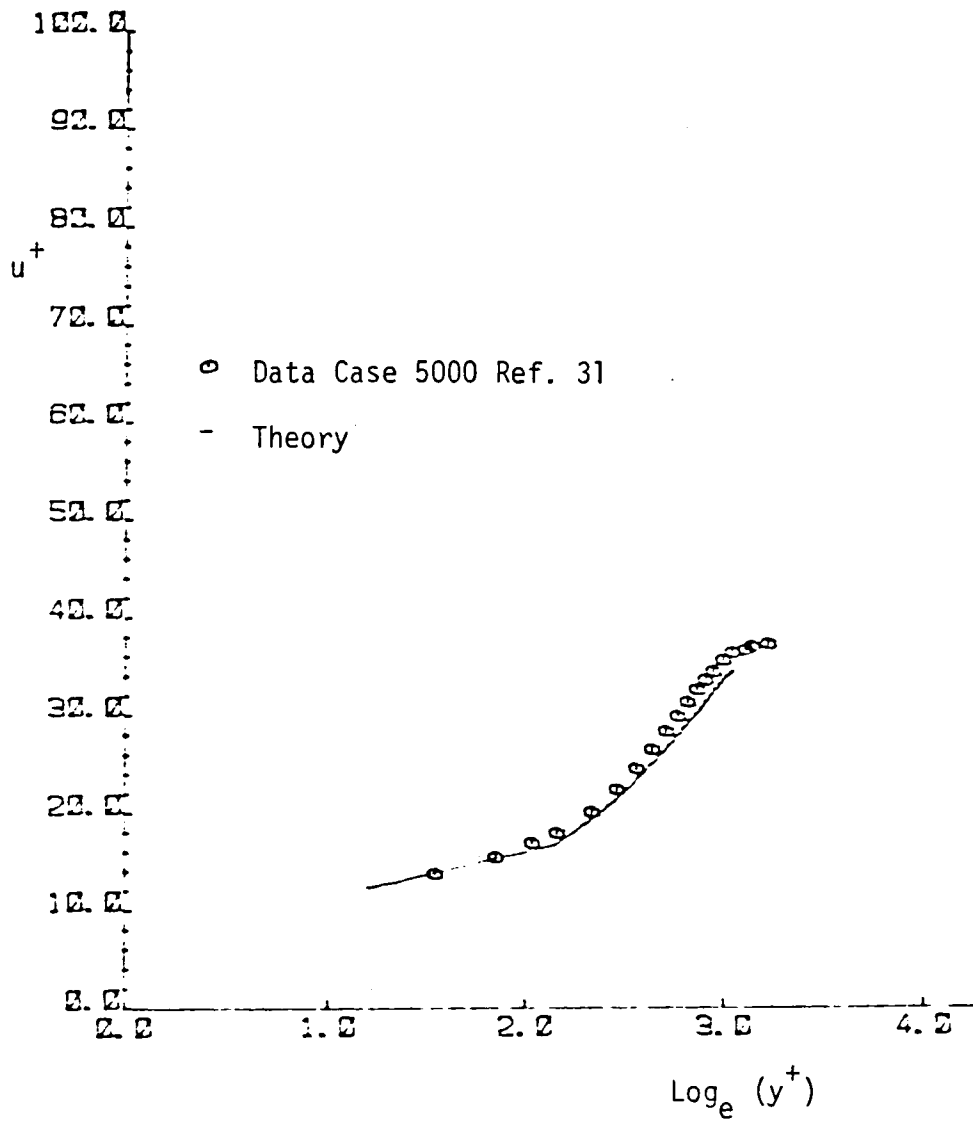


Figure 20. Comparison of Logarithmic Velocity Distribution at $x = 15.049$ Inches

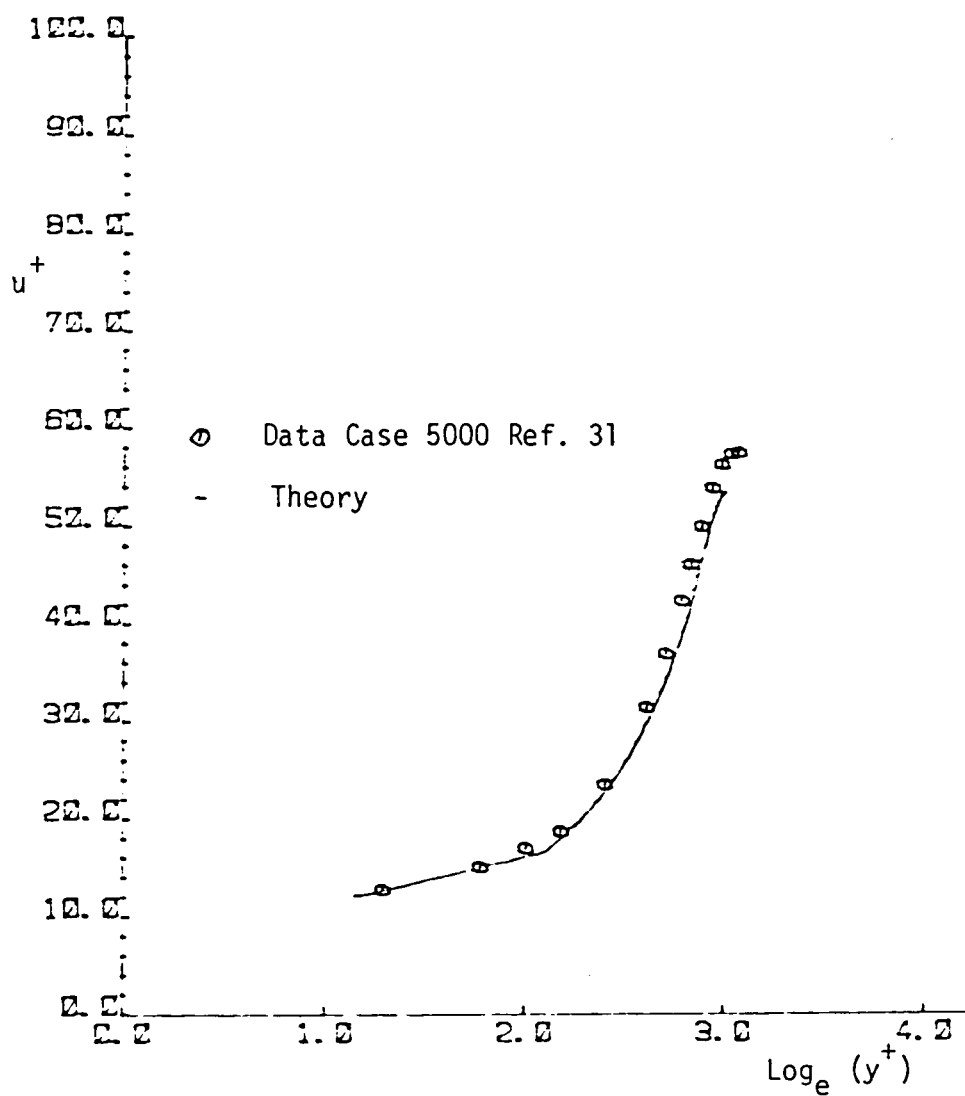


Figure 21. Comparison of Logarithmic Velocity Distribution at $x = 17.249$ Inches

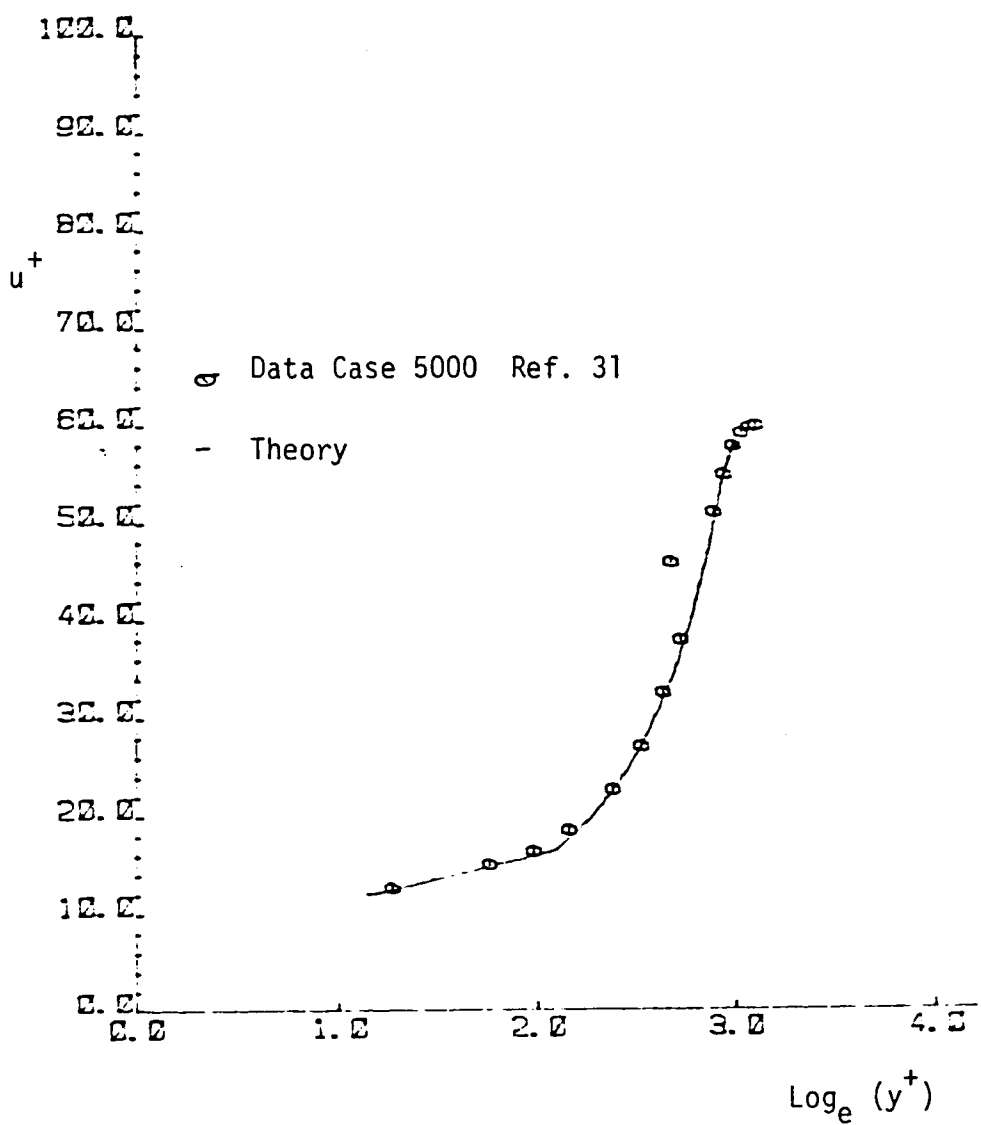


Figure 22. Comparison of Logarithmic Velocity Distribution at $x = 18.449$ Inches

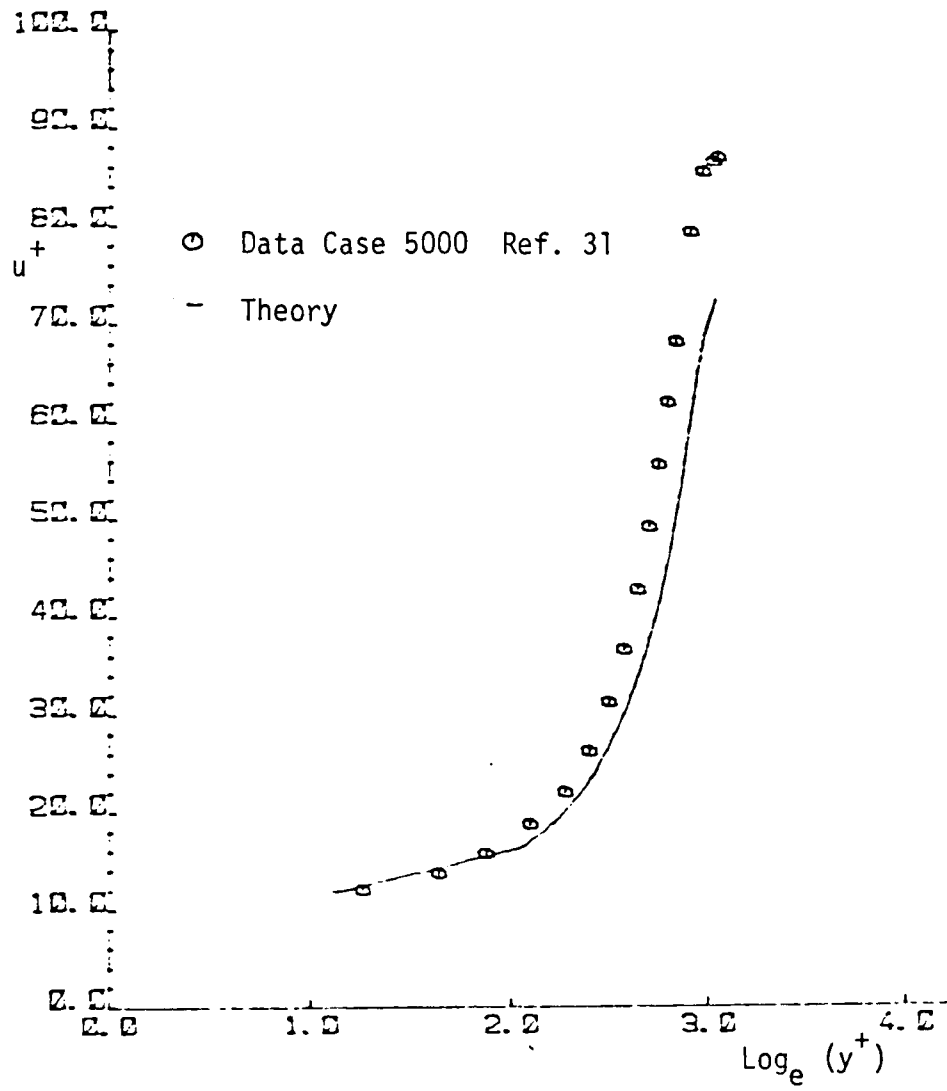


Figure 23. Comparison of Logarithmic Velocity Distribution at $x = 22.749$ Inches

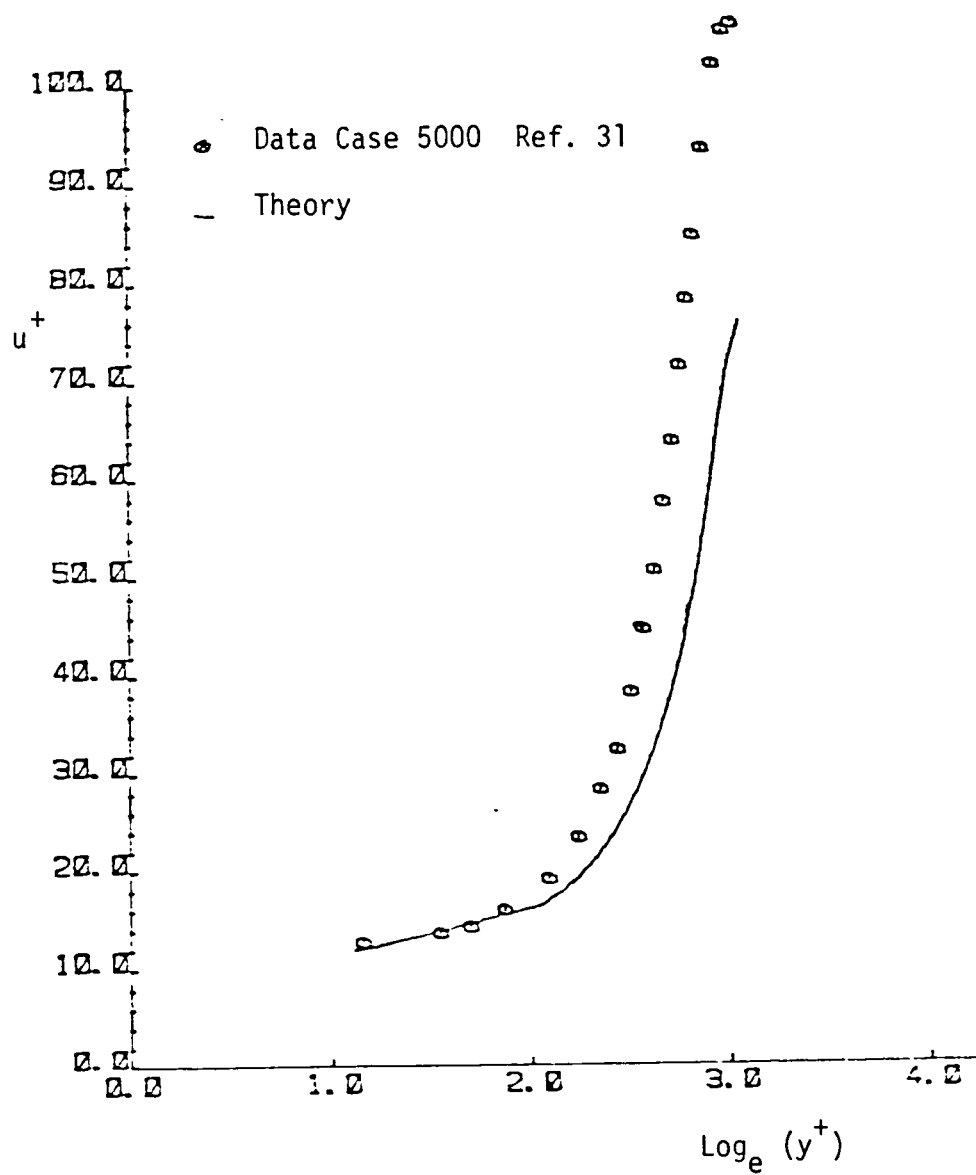


Figure 24. Comparison of Logarithmic Velocity Distribution at $x = 24.149$ Inches

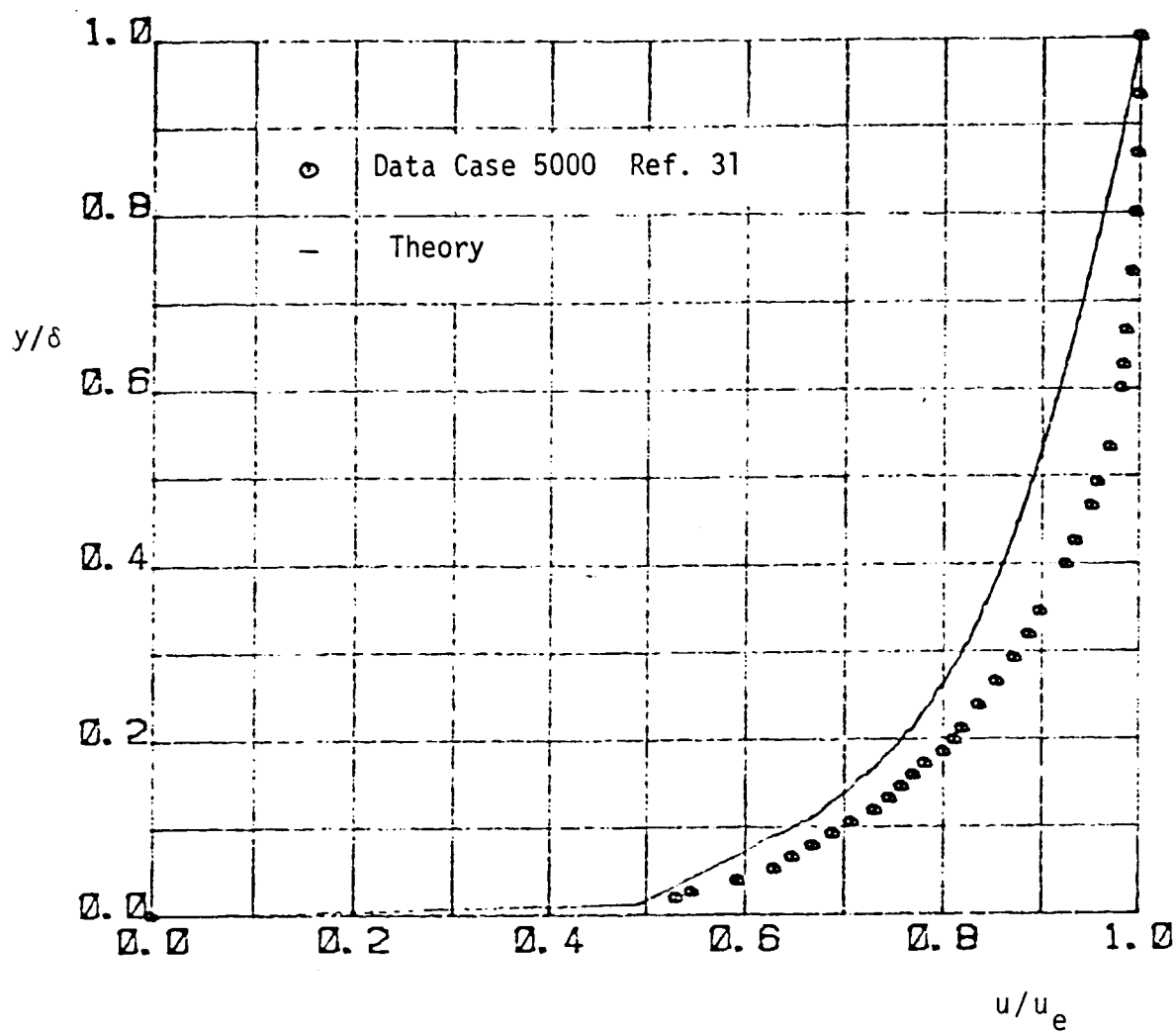


Figure 25. Comparison of Theoretical and Experimental Velocity Profile at $x = 8.29$ Inches

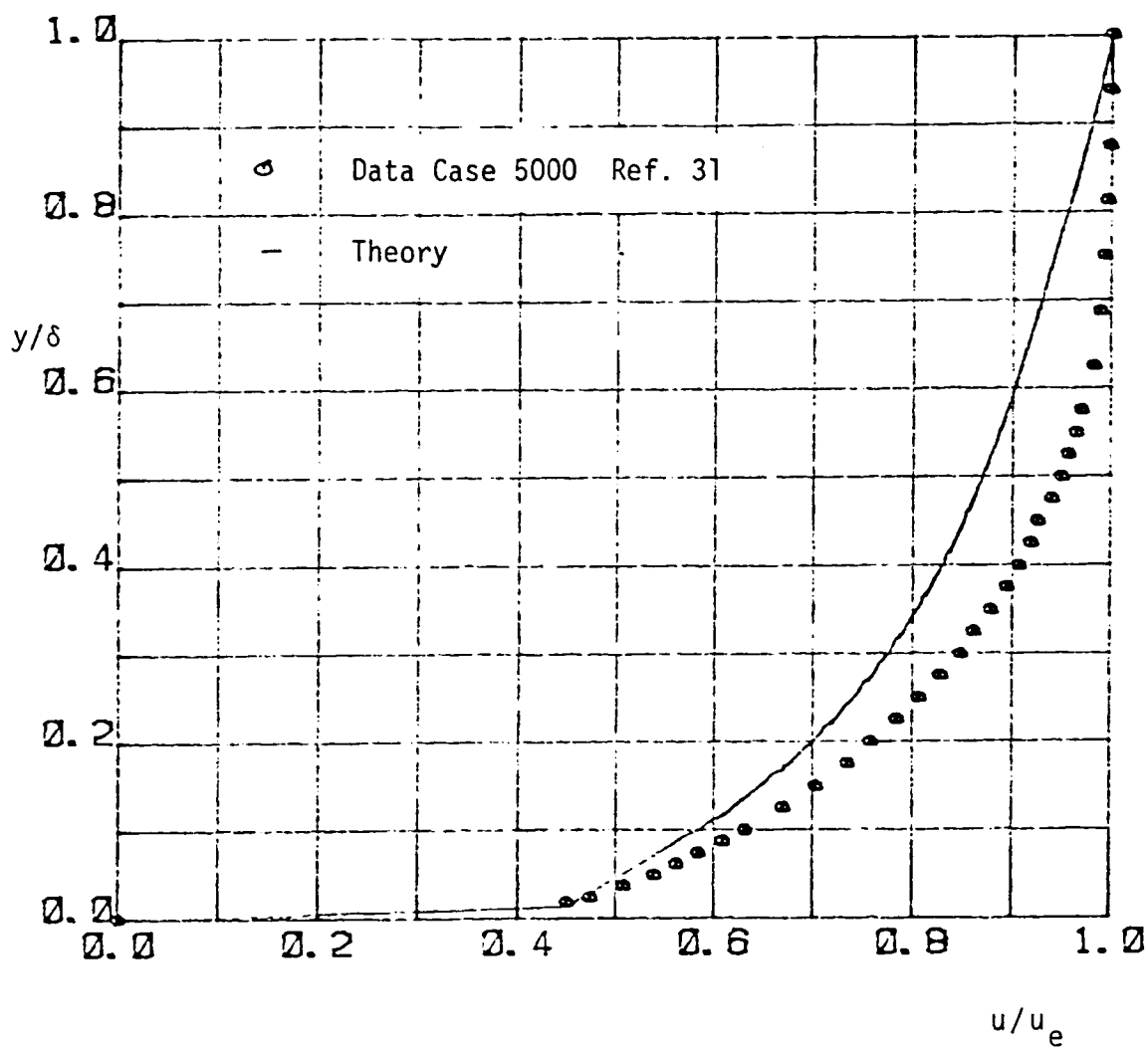


Figure 26. Comparison of Theoretical and Experimental Velocity Profile at $x = 9.29$ Inches

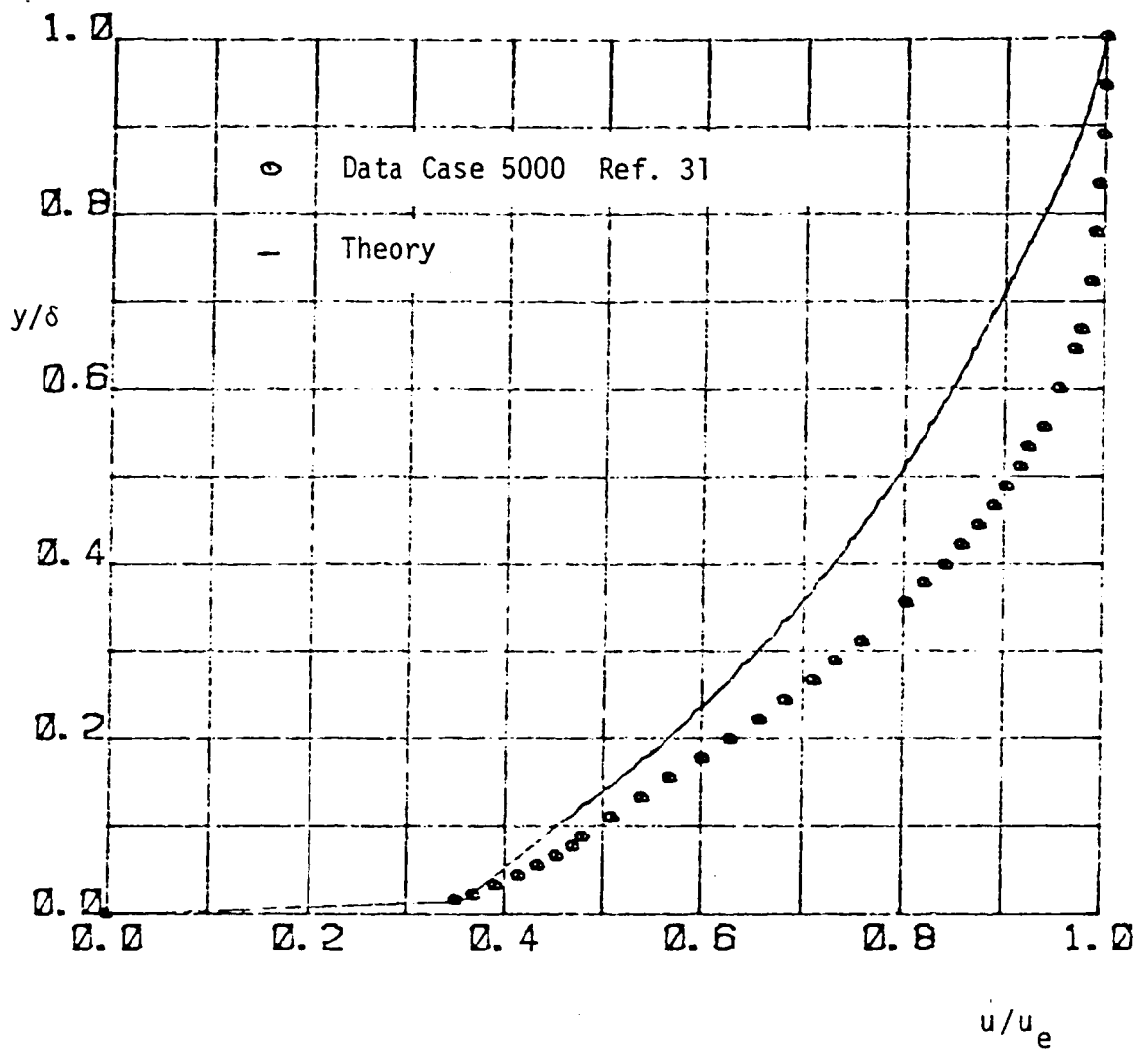


Figure 27. Comparison of Theoretical and Experimental Velocity Profile at $x = 11.79$ Inches

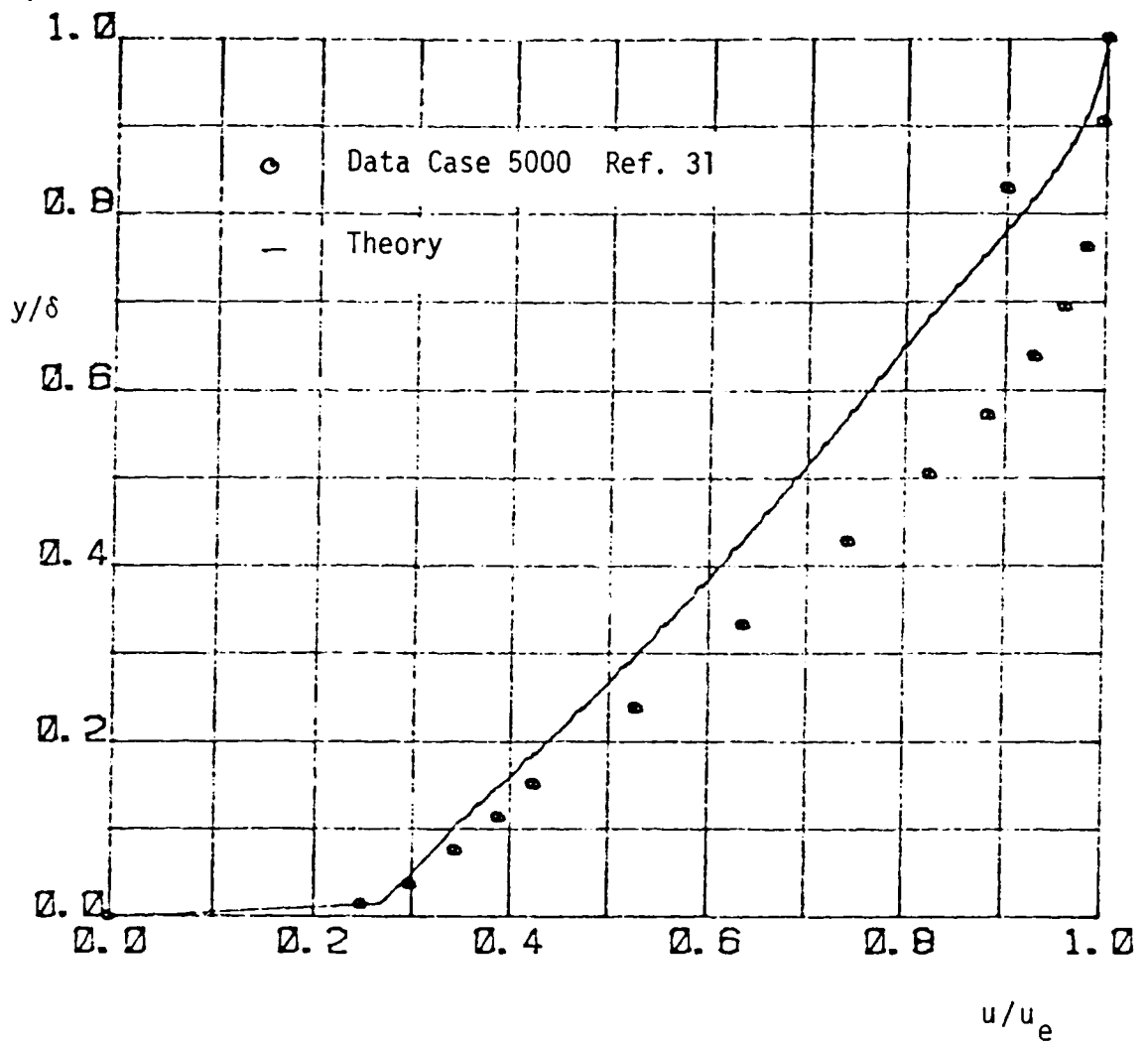


Figure 28. Comparison of Theoretical and Experimental Velocity Profile at $x = 15.04$ Inches

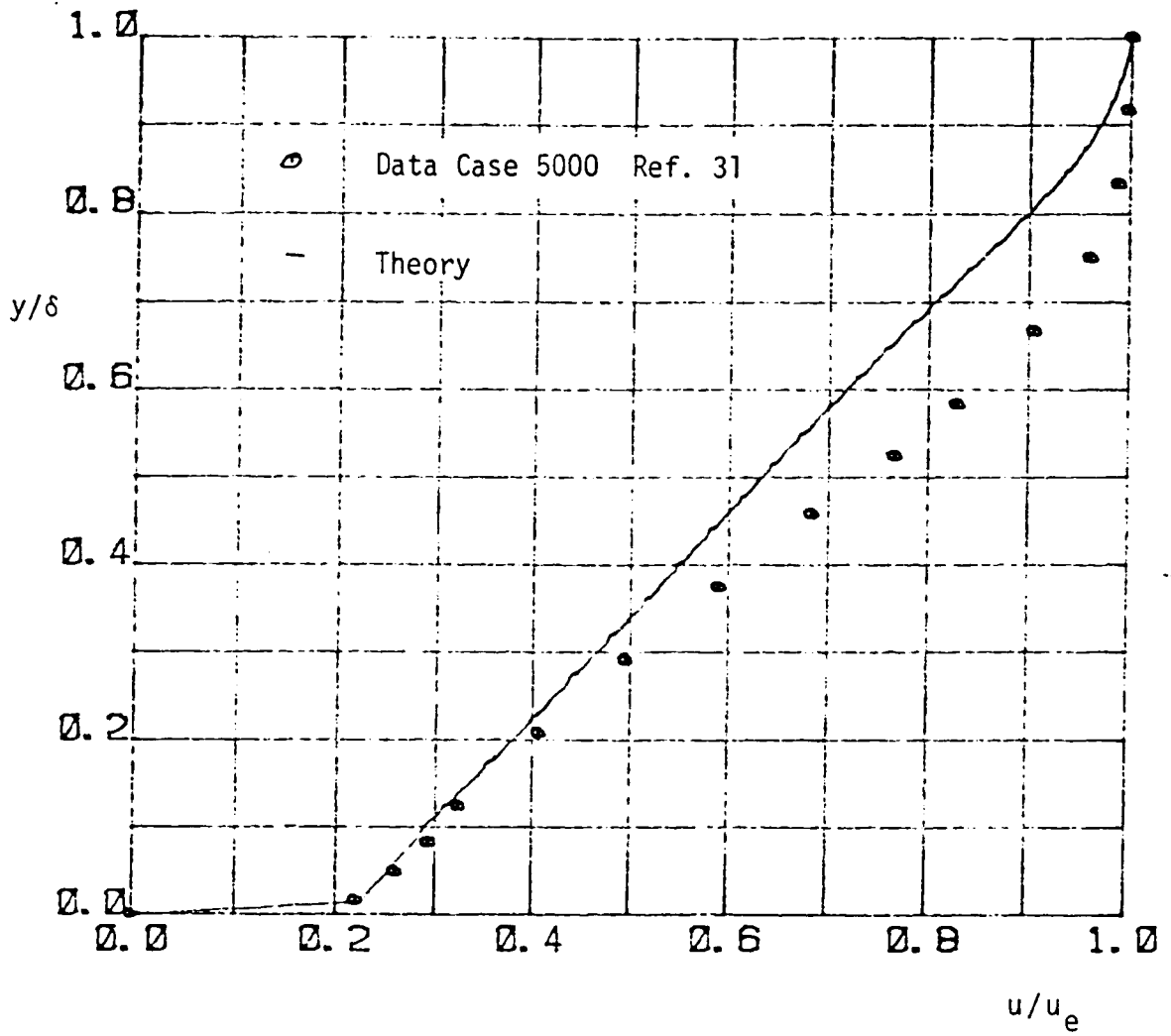


Figure 29. Comparison of Theoretical and Experimental Velocity Profile at $x = 17.24$ Inches

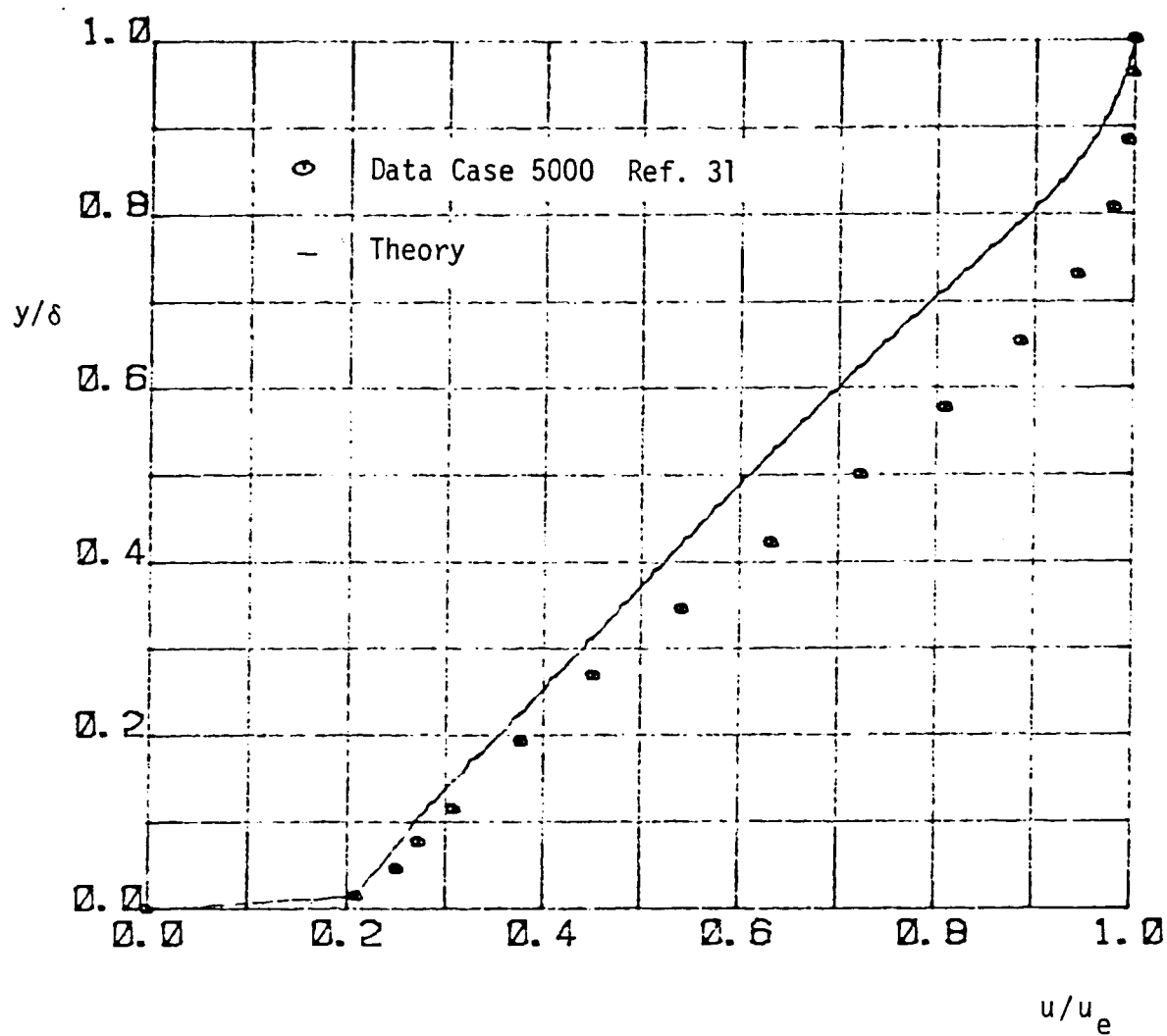


Figure 30. Comparison of Theoretical and Experimental Velocity Profile at $x = 18.449$ Inches

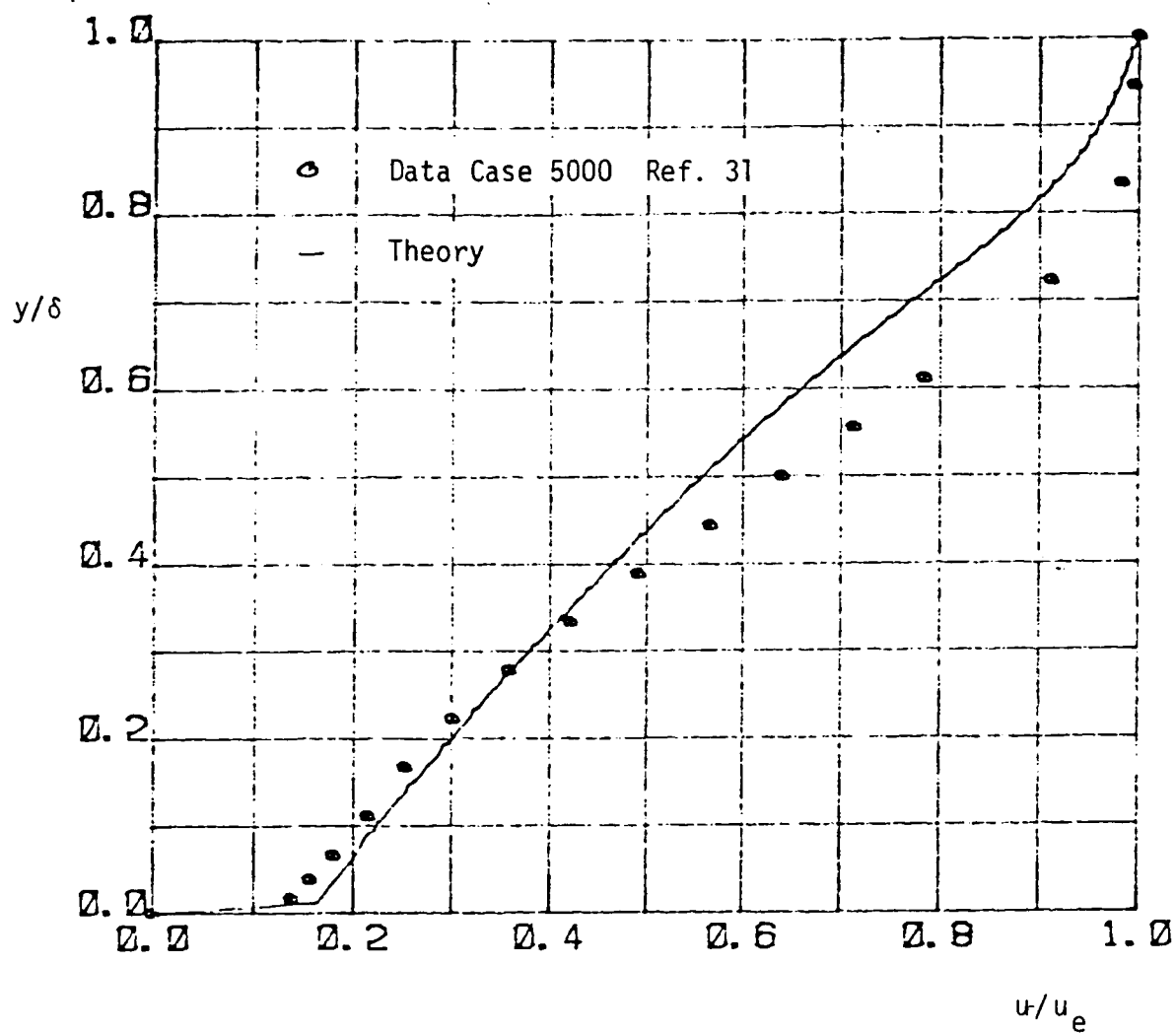


Figure 31. Comparison of Theoretical and Experimental Velocity Profile at $x = 20.749$ Inches

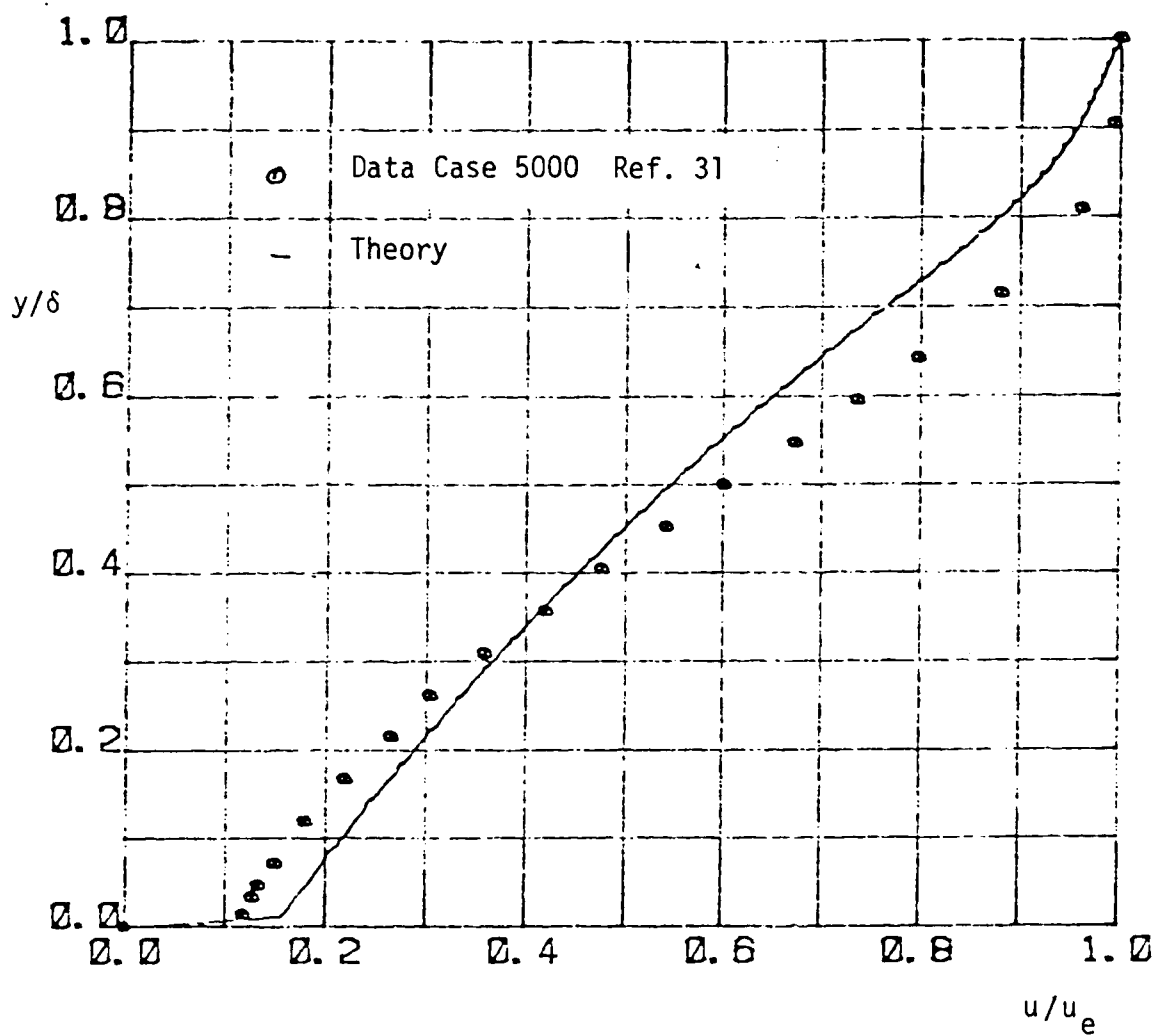


Figure 32. Comparison of Theoretical and Experimental Velocity Profile at $x = 24.149$ Inches

purpose of examining predicted velocity profile shapes, a more sensitive comparison is obtained in physical coordinates.

The friction coefficient C_f and the integral parameters H and θ for the Stanford 5000 Case are compared in Figures 33, 34, and 35 respectively. The prediction maybe considered reasonably good in the upstream side of the diffuser, however, the agreement is less satisfactory in the downstream section.

The apparent lack of agreement in the velocity distributions could be attributed to the mismatch of the exact velocity profile and the generated profile using $1/9$ power law, and to the fact that initial profiles of turbulence kinetic energy were not given experimentally but were derived from Prandtl's Mixing Length distribution and the experimental $\frac{du}{dy}$ profiles.

Performance Prediction for Model Diffuser

An attempt was made to predict the performance of the model experimental diffuser, which is a scaled down version of the diffuser employed in the Coal-Fired Flow Facility. The analysis was carried out using the original code and the modified version of the code. Figure 36 gives the predicted performance map of the model diffuser using the modified procedure. Figure 37 gives the predicted performance using original computation procedure. The main point of interest is that the code set up to use the modified dissipation length scale distribution predicts that the flow separates farther

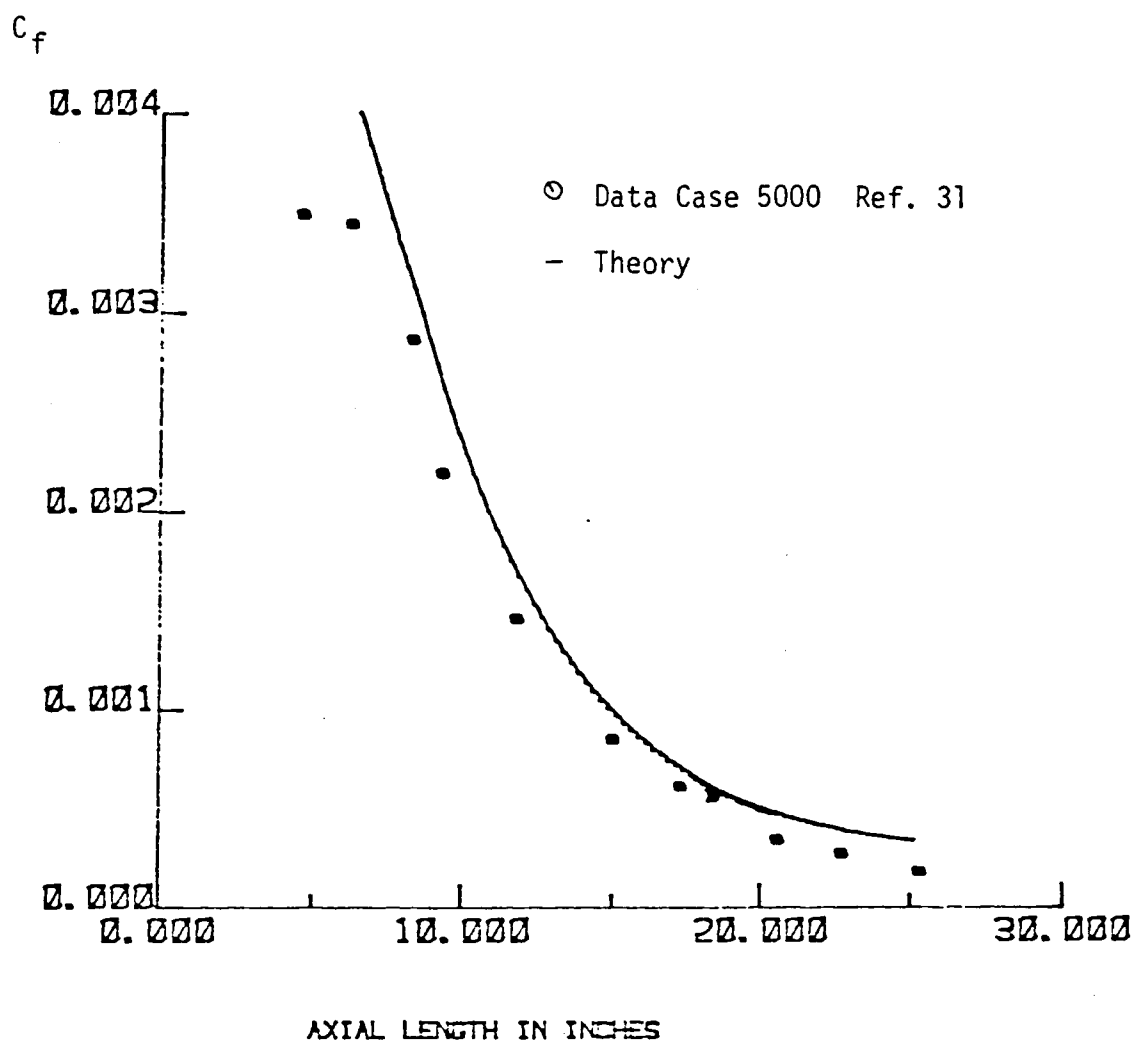


Figure 33. Comparison of Theoretical and Experimental Values of C_f Along Diffuser Length

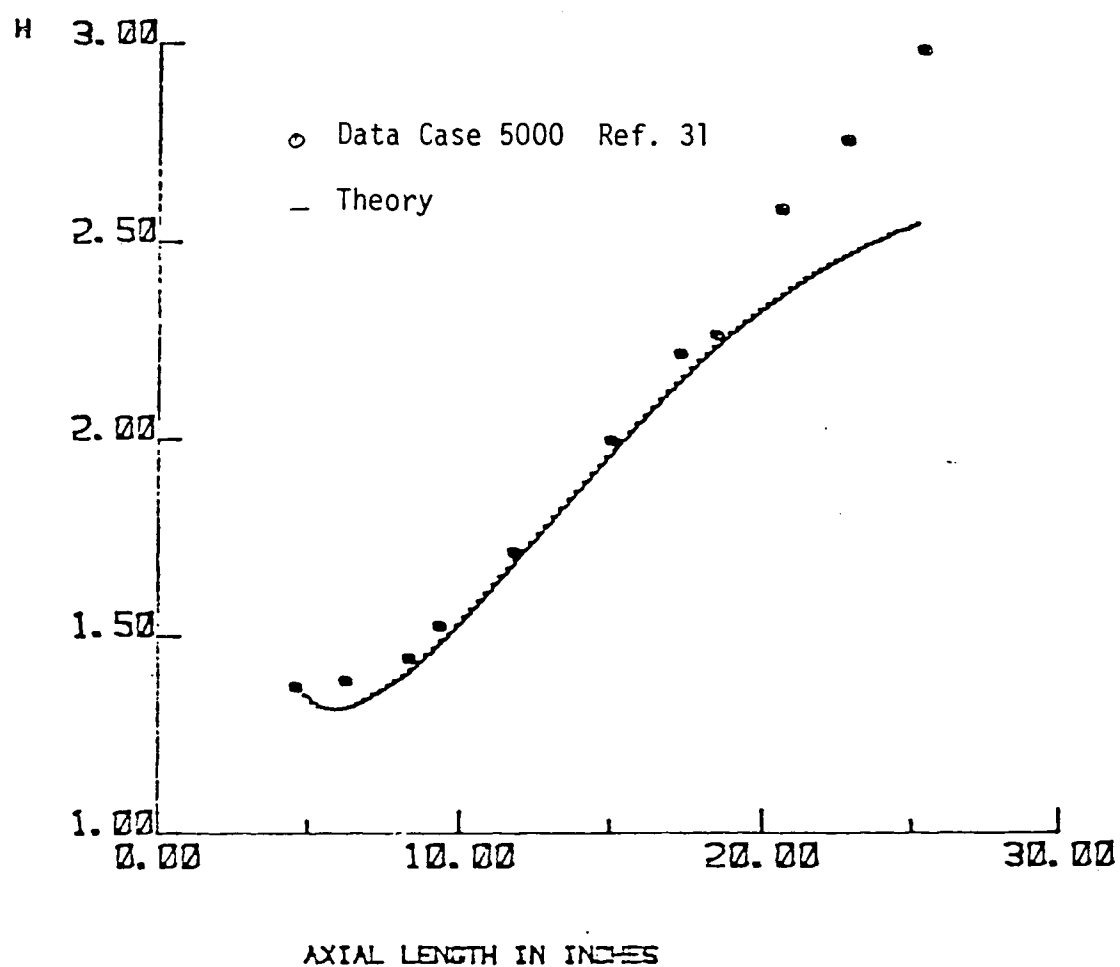


Figure 34. Comparison of Theoretical and Experimental Values of Shape Factor (H) Along Diffuser Length

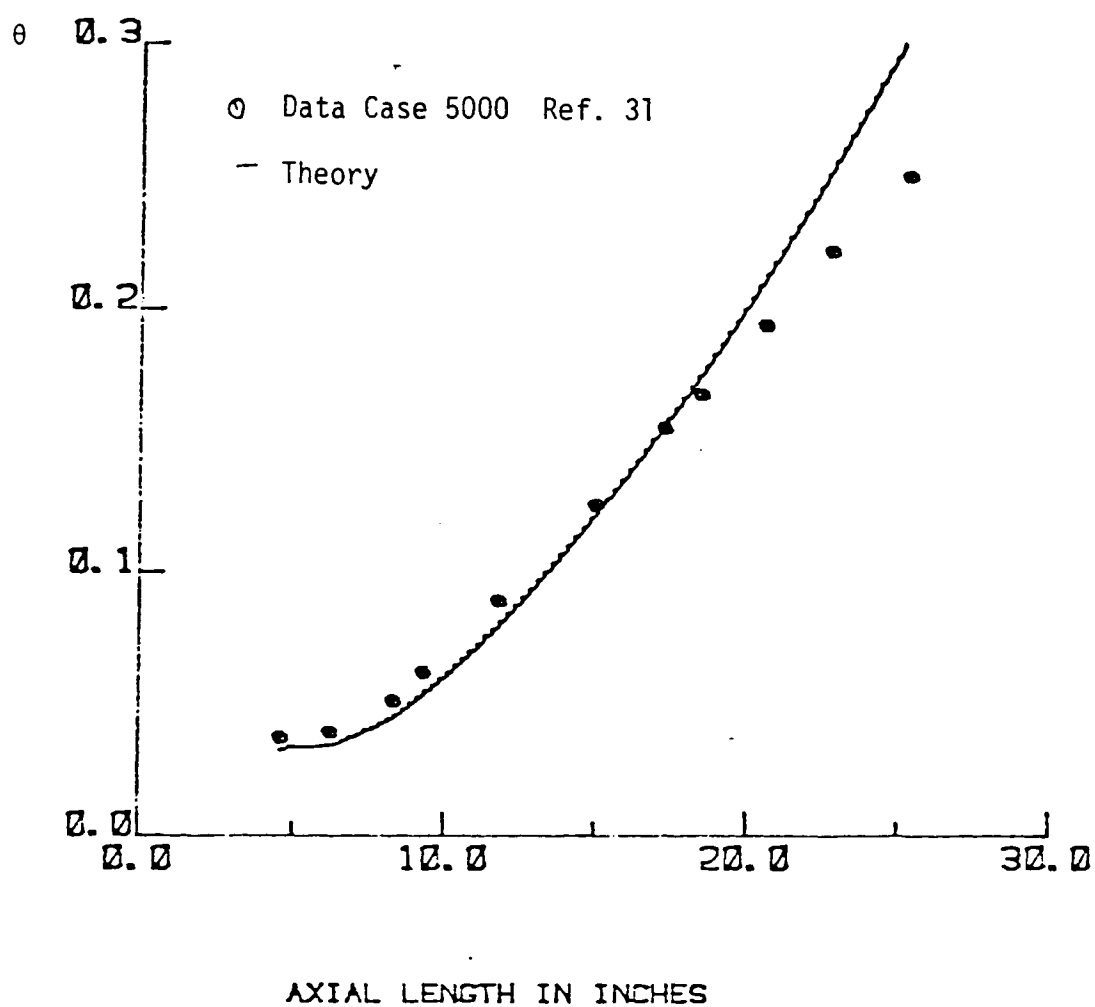


Figure 35. Comparison of Theoretical and Experimental Values of Momentum Thickness (θ) Along Diffuser Length

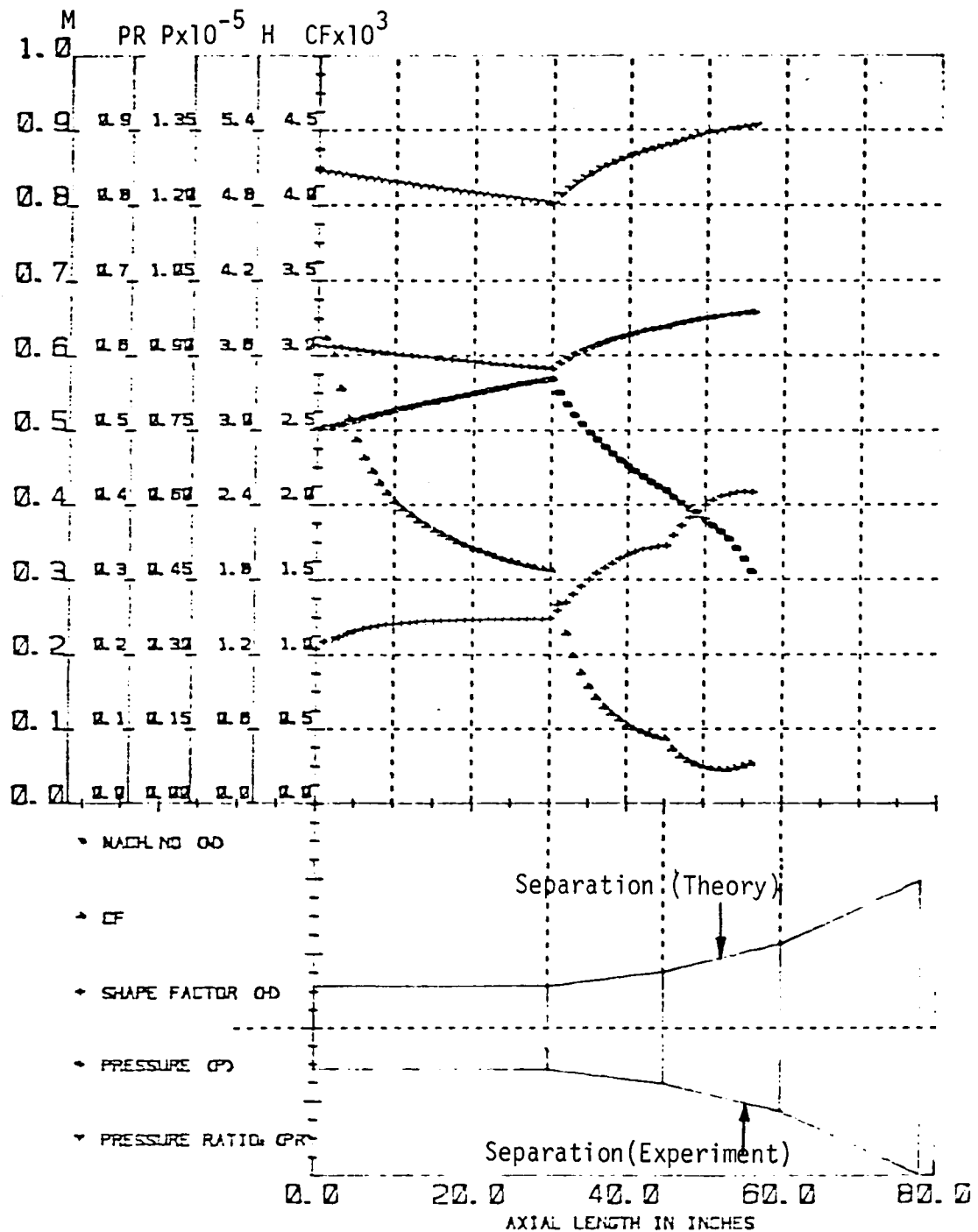


Figure 36. Predicted Diffuser Performance Map for Model Diffuser Using Modified Code

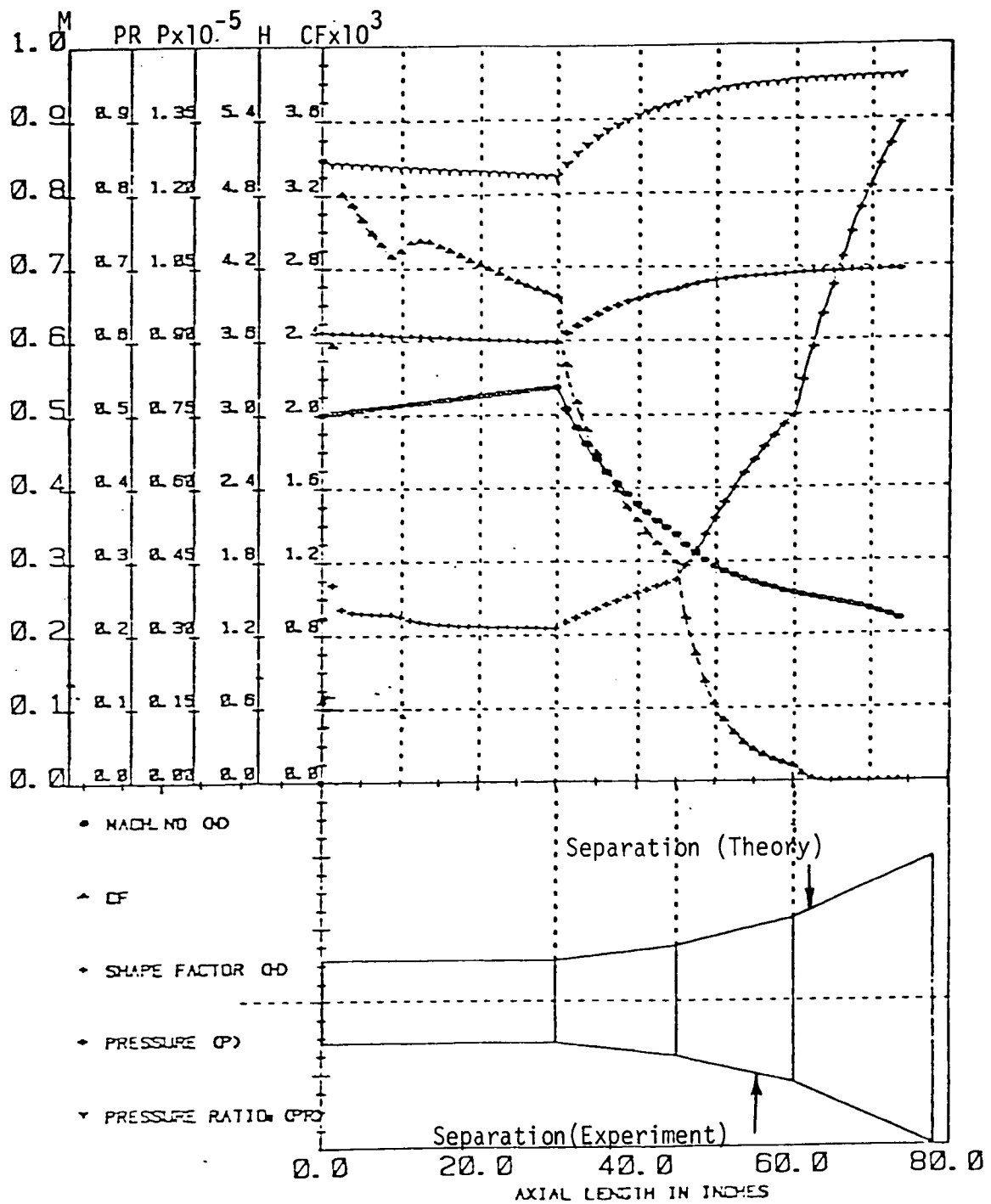


Figure 37. Predicted Diffuser Performance Map for Model Diffuser Using Original Code

upstream than the original version of the code. This result is closer to the experimentally defined separation point as shown on these figures.

The method used to simulate the square cross-section diffuser geometry for the predictions shown in Figure 36 and 37 was Method 1, described in Chapter III. This method sets up a axisymmetric stream tube with a cross-sectional area distribution equal to the actual cross-sectional area distribution (as a function of length along the bulk flow path). Since the other two methods were not evaluated, the "best" way for simulating the diffuser cross-section is not known.

CHAPTER V

CONCLUSIONS AND RECOMMENDATIONS

Conclusions

The law-of-the-wall applies to boundary layers approaching separation provided the Reynold's number is high enough. Thus, for high Reynold's number boundary layers under adverse pressure gradients, the breakdown mode is progressive rather than general.

The present version of the diffuser code predicts the standard data at least as well as the work reported by Harsha and Glassman. But, they provided only two x-locations for comparison and did not specify the inlet conditions that they used precisely enough to permit accurate comparison.

The present version of the computer code predicts separation in the model diffuser farther upstream and hence closer to experiment than the original version.

Not enough work was done in the present study to evaluate which method is best for simulating rectangular diffuser geometry with an axisymmetric code.

Finally, the present study:

1. Has not investigated the thermal law-of-the-wall and surface roughness effects.
2. Has not investigated velocity profile overshoot.
3. Has not investigated different length scale distributions for boundary layers under adverse pressure gradients.

Recommendations for Future Work

Even though provision is made for incorporating wall heat effects, no heat transfer analysis has been made. Further work is required to investigate the thermal law-of-the-wall and wall heat transfer.

The present version of the code has to be further modified to investigate the effects of initial velocity profile overshoot on the separation characteristics of the flow.

Present analysis uses a relatively smooth wall and the wall functions have to be modified to account for surface roughness effects.

Further work is required to evaluate which method is best for simulating rectangular diffuser geometry with an axisymmetric code.

The present version of the code predicts separation close to the experimentally observed point, however, this has to be further tested against experimental data and with various inlet conditions.

In the present analysis only viscosity variation with respect to temperature is taken into account; in the actual MHD case the variation of other thermophysical properties with respect to temperature also has to be considered.

LIST OF REFERENCES

LIST OF REFERENCES

1. Harsha, P. T., and H. N. Glassman. Analysis of Turbulent Unseparated Flow in Subsonic Diffusers," ASME Paper 76-FE-26, 1976.
2. Kline, S. J., D. E. Abbot, and R. W. Fox. "Optimum Design of Straight Walled Diffusers," Journal of Basic Engineering, September, 1959.
3. Runstadler, Jr., P. W. and R. C. Dean. "Straight Channel Diffuser Performance at High Inlet Mach Numbers," Journal of Basic Engineering, September, 1969.
4. Reneau, L. R., J. P. Johnston and S. J. Kline. "Performance and Design of Straight Two-Dimensional Diffusers," Report PD-8, Department of Mechanical Engineering, Stanford University, 1964.
5. Maxwell, C. D., E. D. Doss, B. P. Cory, and D. A. Oliver. "Consideration of Three-Dimensional Effects in MHD Power Generators," Symposium on Engineering Aspects of MHD 15th. The University of Pennsylvania, 1976, p. 151-154.
6. Rankin, R. R., S. A. Self, and R. H. Eustis. "A Study of the MHD Insulating Wall Boundary Layer," Symposium on Engineering Aspects of MHD, 16th, University of Pittsburgh, 1977, X81-X811.
7. Greywall, Masesh S., and J. Martin Smith. "The Effect of Velocity Overshoot on the Performance of Magnetohydrodynamic Subsonic Diffusers," AIAA Paper 80-0251, 1980, AIAA 18th Aerospace Sciences Meeting, Pasadena, California, Jan. 14-16, 1980.
8. White, Frank M. Viscous Fluid Flow, McGraw-Hill, 1974.
9. Granville, P. S. "A Modified Law-of-the-Wake for Turbulent Shear Layers," Transactions of ASME Series II, June, 1975.
10. Rasmussen, M. L. "On Compressible Turbulent Boundary Layers in the Presence of Favourable Pressure Gradients," Journal of Fluids Engineering, ASME Paper 75-WA/HT-53, 1975.
11. Dean, R. B. "A Simple Formula for the Complete Velocity Profile in a Turbulent Boundary Layer," Journal of Fluids Engineering, Transactions of ASME, September, 1976.

12. McD Galbraith, R. A., S. Sjolander, and M. R. Head. "Mixing Length in the Wall Region of Turbulent Boundary Layers. Aeronautical Quarterly, May 1977, pp. 97-100.
13. Launder, B. E. "Progress in the Modelling of Turbulent Transport," Von Karman Institute Lecture Series (LS)-76, March, 1975.
14. Pimenta, M. M., R. J. Moffat, and W. M. Kays. "The Structure of a Boundary Layer on a Rough Wall with Blowing and Heat Transfer," Journal of Heat Transfer, May, 1979, pp. 193-198.
15. Lewis, M. J. "Optimising the Thermohydraulic Performance of Rough Surfaces," Int. Journal of Heat and Mass Transfer, Volume 18, 1975, pp. 1243-1248.
16. Perry, A. E., W. H. Schofield, and P. N. Joubert. "Rough Wall Turbulent Boundary Layers," Journal of Fluid Mechanics, Volume 37, 1969.
17. Kays, W. M., and M. E. Crawford. Convective Heat and Mass Transfer, Second Edition, McGraw Hill, 1980.
18. Dipprey, D. F., and R. H. Sabersky. "Heat and Momentum Transfer in Smooth and Rough Tubes at Various Prandtl Numbers," International Journal of Heat and Mass Transfer, Volume 6, 1963, pp. 329-353.
19. Blom, J. "Experimental Determination of the Turbulent Prandtl Number in a Developing Temperature Boundary Layer," Heat Transfer, 1970, (Paper Presented at the Fourth International Heat Transfer Conference at Pans-Versailles, 1970).
20. Launder, B. E., and D. B. Spalding. "The Numerical Computation of Turbulent Flows," Computational Methods in Applied Mechanics and Engineering (3), 1974, pp. 269-289.
21. Lee, S. C., and P. T. Harsha. "The Use of Turbulent Kinetic Energy in Free Mixing Studies," AIAA Journal, Volume 8, No. 6, June 1970, p. 1026-1032.
22. Harsha, P. T. "A General Analysis of Free Turbulent Mixing, AEDC-TR-73-177, May, 1974.
23. Launder, B. E., and D. B. Spalding. Lectures in Mathematical Models of Turbulence. Academic Press, London, 1972.

24. Hinze, J. O. Turbulence, Second Edition, McGraw Hill, 1975.
25. Pantankar, S. V., and D. B. Spalding. Heat and Mass Transfer in Boundary Layers. Intertext Books, London, 1970.
26. Sajben, M., J. C. Kroutil, A. V. Sedrick, and G. H. Hoffman. "Experiments on Conical Diffusers with Distorted Inflow," AIAA Paper No. 74-529, 1974.
27. Brogen, T. R., J. J. Idzorek, and D. Swallom. "Subsonic Diffusers for MHD Generators," Fuils Engineering in Advanced Energy Systems. Proceedings of the Winner Annual Meeting, San Francisco, California, Dec. 1978, ASME p. 125-138.
28. Michio Mishioka and Shusuke Iida. "Mixing Length Derived from Karman's Similarity Hypothesis," Aeronautical Quarterly, February, 1973.
29. Patel, V. C. "A Unified View of the Law-of-the-Wall Using Mixing Length Theory," Aeronautical Quarterly, Volume XXIV, February, 1973, pp. 55-70.
30. Young, A. D., and G. L. Green. "Tests of High Speed Flow in Diffusers of Rectangular Cross-Section," Aeronautical Research Council Technical Report 2201, 1944.
31. Coles, D. E. and E. A. Hirst. "Proceedings - Computations of Turbulent Boundary Layers - 1968 AFOSR.1FP Stanford Conference, Volume II, Compiled Data, Therophysics Division, Department of Mechanical Engineering, Stanford University, 1969.
32. Bradshaw, P., D. H. Ferriss, and N. P. Atwell. "Calculation of Boundary Layer Development Using the Turbulent Energy Equation," Journal of Fluid Mechanics, 28-593, 1967.
33. Written Communication from Dr. Roy J. Schulz. Assistant Professor, Mechanical/Aerospace Engineering, UTSI.
34. Roy, G. D. "On Pressure and Heat Flux Distribution Along Magnetohydrodynamic Generator Channel Diffuser Systems, Ph.D. Dissertation, UT Knoxville, March, 1977.
35. Written Communication from Dr. Ahmad Vakili. Gas Dynamics Division, UTSI.

APPENDIX

BOUNDARY LAYER INTEGRAL PARAMETERS

Displacement Thickness δ^*

$$\delta^* = \int_0^{\infty} \left(1 - \frac{\rho u}{\rho_{\infty} u_{\infty}}\right) dy$$

Momentum Thickness θ

$$\theta = \int_0^{\infty} \frac{\rho u}{\rho_{\infty} u_{\infty}} \left(1 - \frac{u}{u_{\infty}}\right) dy$$

Shape Factor H

$$H = \frac{\delta^*}{\theta}$$

Defect Thickness Δ

$$\Delta = \int_0^{\infty} \frac{u_{\infty} - u}{u_{\tau}} dy$$

Dissipation Integral D

$$D = \int_0^{\infty} \frac{\tau}{\tau_w} \frac{\partial(u/u_{\infty})}{\partial(y/\theta)} d(y/\theta)$$

VITA

Vidyasagaran, N.S. was born in Chowghat, India on May 10, 1949. He attended elementary school in Manathala. He passed the Secondary School Leaving Certificate Examination in 1964. He entered University of Kerala and passed the Pre Degree Examination in 1966. In the same year, he was awarded the National Science Talent Search Scholarship and Certificate of Merit. In 1966, he entered Engineering College, Trichur in Kerala University to pursue Bachelor of Science in Engineering with a major in Mechanical Engineering. He received BS (Engineering) degree with honours in May, 1970. After graduation, he joined the Propulsion Division of Vikram Sarabhai Space Center (A unit of Indian Space Research Organization), in September, 1970. As propulsion engineer, he was engaged in the development of Monopropellant Hydrazine Propulsion Systems until August 1979. In September 1979, he joined The University of Tennessee Space Institute and completed the requirements for the Master of Science degree in December, 1981.