

FLOW AND COMBUSTION CHARACTERISTICS
OF A LIQUID-FUELLED COMBUSTOR

SIU-KEI CHOW

B. Eng. (Hons.), ACGI

THESIS SUMMITTED FOR THE DEGREE OF
DOCTOR OF PHILOSOPHY
IN THE UNIVERSITY OF LONDON
AND FOR THE
DIPLOMA OF IMPERIAL COLLEGE

IMPERIAL COLLEGE OF SCIENCE,
TECHNOLOGY AND MEDICINE
(DEPARTMENT OF MECHANICAL ENGINEERING)

JUNE, 1991

"Logic is bound to the condition: assume there are identical cases. In fact, to make possible logical thinking and inferences, this condition must first be treated fictitiously as fulfilled. That is: the will to logical truth can be carried through only after a fundamental falsification of all events is assumed. From which it follows that a drive rules here that is capable of employing both means, firstly falsification, then the implementation of its own point of view: logic does not spring from will to truth."

FRIEDRICH NIETZSCHE, 1844-1900

ABSTRACT

Flow and combustion characteristics inside models of an annular combustor have been studied. The combustors were operated at a preheated air temperature of 515K and the liquid fuel was introduced through a T-vaporiser. Velocity measurements were obtained with laser-Doppler velocimetry, temperature measurements with digitally-compensated fine-wire thermocouples, and species concentrations of CO, CO₂, O₂, H₂ and UHC sampled through a quartz microprobe and a water-cooled probe with infra-red analysers, paramagnetic analyser, gas chromatograph and flame ionisation detector.

Experiments were carried out in a model combustor without an exit nozzle with air-fuel ratios of 26 and 43, corresponding to those of take-off and ground-idle conditions respectively. The results, when compared with previous measurements using gaseous fuel, showed that the use of liquid fuel led to decreases in combustion efficiency from 94.2 to 92.3% and 97.6 to 96.2% at the take-off and ground-idle conditions respectively and to corresponding decreases in pattern factor from 0.37 to 0.19 and 0.48 to 0.37. The flowfields inside the combustor were simulated by a computer code involving finite difference solution of conservation equations including a two-equation turbulence model. The turbulence/chemistry interaction was modelled by a β -function probability density function of a single conserved scalar. A comprehensive parametric study was carried out to investigate the effects of boundary conditions for air jets, slots and vaporiser in isothermal flows. For reacting flow calculations, the equilibrium combustion model resulted in high primary zone concentrations of CO and H₂, with maximum predicted values of 9% and 16% respectively, compared with 2.5% and 1.75% in measured values.

A flamelet model gave a better representation with maximum values of 4.5% for CO and 3.5% for H₂.

A separate rig was used to study the flow characteristics of the reverse flow nozzle in more detail. This perspex model provided a more uniform entry profile to the nozzle and better optical access. At inlet to the bend, the streamwise velocity profiles showed a sharp peak close to the inner surface with a maximum value of about 1.3 times that of the bulk mean velocity and crossflow velocities towards the centre of the bend were observed with maximum magnitude of about 25% of the bulk mean velocity. At further downstream stations, the profiles of the streamwise velocity remained unchanged and became more evenly distributed towards the exit. The geometry was reproduced numerically, using a general curvilinear co-ordinate system based on a non-orthogonal boundary conforming transformation and distributions of streamwise velocity profile were predicted to an accuracy within 7% in the core flow region but the near wall profiles were less well predicted due to the inability of the k- ϵ turbulence model to describe secondary flows.

Temperature and concentration values were measured at the exit of the model combustor with a curved exit nozzle, with air-fuel ratios corresponding to the take-off and ground-idle conditions. For an air-fuel ratio of 60, corresponding to the ground-idle condition, the averaged CO and UHC concentrations were about 0.1 and 0.005% at the exit of the curved exit nozzle, and led to emission indices of 50 and 5 respectively. The temperature profiles showed a more uniform distribution at the exit and hence an improved pattern factor. With an air-fuel ratio of 36, the emission indices increased to 68 and 6 for CO and UHC respectively.

ACKNOWLEDGEMENTS

I would like to express my gratitude to all those who provided assistance and encouragement during the course of my research and for the preparation of this thesis.

First of all, I wish to thank my supervisor, Professor J. H. Whitelaw, who provided me with sound guidance, valuable criticism and suggestions throughout the course of this work.

I am especially grateful to Dr. W. P. Jones and Dr. J. J. McGuirk for their constant interest in my work and their generous assistance on the numerical calculations and Dr. R. P. Linstedt for making available the flamelet data. I am also obliged to Professor M. Senda for his support on the experimental work. The work has benefitted from numerous discussions and advice with my colleagues at Imperial College and thanks are due to Dr. M. Dianat, Mr. A. Tober, Mr. J. Alvarez, Mr. S. Baker, Dr. D. Tse and Mr. S. Yu. I am also pleased to acknowledge the help of Mr. J. Gardiner of Rolls Royce, Leavesden in the design of the combustor.

I thank Mr. W. Crew, Mr. H. Avis and Mr. J. Hewison for the help in setting up the experimental apparatus.

I welcome the opportunity to acknowledge the financial support provided by the Procurement Executive of the Ministry of Defence.

Finally, I deeply appreciate the love and support of my family and to them I dedicate this thesis.

CONTENT

	<u>page</u>
ABSTRACT	3
ACKNOWLEDGEMENTS	5
CONTENT	6
LIST OF TABLES	9
LIST OF FIGURES	10
NOMENCLATURES	18
CHAPTER 1 INTRODUCTION	22
1.1 Introduction	22
1.2 Previous Work	25
1.2.1 Bend Flow	26
1.2.2 Gas Turbine Combustor	30
1.2.3 Numerical Modelling	34
1.3 Present Contribution	38
1.4 Layout of the Thesis	40
CHAPTER 2 EXPERIMENTAL TECHNIQUES	59
2.1 Introduction	59
2.2 Velocity Measurements	61
2.2.1 Pressure Probe	61
2.2.2 Laser-Doppler Velocimetry	62
2.3 Temperature Measurements	65
2.4 Species Concentrations Measurements	67
CHAPTER 3 CALCULATION PROCEDURE	78
3.1 Introduction	78
3.2 Description of the Equations	79
3.2.1 Exact Equations	79
3.2.2 Turbulence Model	81
3.2.3 Combustion Model	83
3.3 Methods for Calculations	87
3.3.1 Finite Difference Discretisation	87

3.3.2 Boundary Conditions	92
3.3.3 Mesh Generations	96
CHAPTER 4 FLOW IN A CURVED DUCT	101
4.1 Introduction	101
4.2 Flow Configuration	102
4.3 Details of Calculations	103
4.4 Results	104
4.4.1 Measurements	104
4.4.2 Calculations	109
4.5 Discussion	113
4.6 Conclusions	116
CHAPTER 5 FLOW AND COMBUSTION CHARACTERISTICS OF A MODEL ANNULAR COMBUSTOR	157
5.1 Introduction	157
5.2 Flow Configuration	160
5.3 Details of Calculations	161
5.4 Results	163
5.4.1 Measurements	163
a. Temperature	165
b. Species Concentrations	166
5.4.2 Calculations	168
a. Isothermal Flow	168
b. Combusting Flow	174
5.5 Discussion	177
5.6 Conclusions	184
CHAPTER 6 FLOW AND COMBUSTION CHARACTERISTICS OF A MODEL ANNULAR COMBUSTOR WITH CURVED EXIT NOZZLE	252
6.1 Introduction	252
6.2 Flow Configuration	253
6.3 Results	254
6.3.1 Velocity	255
a. Isothermal Flow	255
b. Combusting Flow	257
6.3.2 Temperature	258
6.3.3 Species Concentrations	259

6.4 Discussion	260
6.5 Conclusions	262
CHAPTER 7 CLOSURE	282
7.1 Conclusions	282
7.2 Further Work	284
REFERENCE	286

LIST OF TABLES

	<u>page</u>
Chapter 1	
Table 1.1 Research in Bend Flow	42
Table 1.2 Experiments in Gas Turbine Combustors	45
Table 1.3 Numerical Modelling of Gas Turbine Combustors	51
Chapter 2	
Table 2.1 Characteristics of the Optical System	71
Chapter 5	
Table 5.1 Geometric Details of Gem Combustor	187
Table 5.2 Properties of Aviation Grade Kerosene	188
Table 5.3 Air Distributions in Gem-60 Combustor	189
Table 5.4 Performance of the Model Combustor	189
Chapter 6	
Table 6.1 Geometric Details of Combustor	264
Table 6.2 Performance of the Combustor	264

LIST OF FIGURES

	<u>page</u>
Chapter 1	
Figure 1.1 Three Types of Combustor	56
Figure 1.2 Reverse-Flow Annular Combustor	57
Figure 1.3 Gem-60 Combustor Geometry	58
Chapter 2	
Figure 2.1 Dimensions of the 5-hole Pressure Probe	72
Figure 2.2 Diagram of the Laser-Doppler Velocimetry	73
Figure 2.3 Dimensions of the Thermocouple Probe	74
Figure 2.4 Schematic of the Thermocouple System	74
Figure 2.5 Dimensions of the Water-Cooled Probe	75
Figure 2.6 Dimensions of the Quartz Microprobe	75
Figure 2.7 Schematic of the Sampling System of the Quartz Microprobe	76
Figure 2.8 Schematic of the Sampling System of the Water- Cooled Probe	77
Chapter 3	
Figure 3.1 Methods of Grid Arrangement	100
Chapter 4	
Figure 4.1 Flow Arrangement	118
Figure 4.2 Geometry of Curved Nozzle	118
Figure 4.3 Finite Difference Meshes used in the Calculations	119
Figure 4.4 Co-ordinates System used for the Curved Nozzle	120

Figure 4.5	Measurements of Mean Streamwise Velocities Upstream of Entry to Curved Nozzle for $Re_{inlet}=15,000$	121
Figure 4.6	Measurements of Mean and rms Streamwise Velocity at 1 Hydraulic Diameter Upstream of Entry to Curved Nozzle for $Re_{inlet}=15,000$	122
Figure 4.7	Measurements of Mean and rms Velocity at Entry to Nozzle (0° Plane) for $Re_{inlet}=15,000$	123
Figure 4.8	Measurements of Mean and rms Velocity at 45° Plane for $Re_{inlet}=15,000$	125
Figure 4.9	Measurements of Mean and rms Velocity at 90° Plane for $Re_{inlet}=15,000$	127
Figure 4.10	Measurements of Mean and rms Velocity at 135° Plane for $Re_{inlet}=15,000$	129
Figure 4.11	Measurements of Mean and rms Velocity at 180° Plane for $Re_{inlet}=15,000$	131
Figure 4.12	Measurements of Mean Velocity at Exit of Nozzle for $Re_{inlet}=15,000$	133
Figure 4.13	Vector Plot of Measured Velocity at Exit of Nozzle for $Re_{inlet}=15,000$	134
Figure 4.14	Measured Pressure Distribution at $Z^*=0$ for $Re_{inlet}=15,000$	134
Figure 4.15	Measured Pressure Distribution at $Z^*=1$ for $Re_{inlet}=15,000$	135
Figure 4.16	Measurements of Mean and rms Streamwise Velocity at 1 Hydraulic Diameter Upstream of Entry to Curved Nozzle for $Re_{inlet}=40,000$	136
Figure 4.17	Measurements of Mean and rms Velocity at Entry to Nozzle (0° Plane) for $Re_{inlet}=40,000$	137

Figure 4.18 Measurements of Mean and rms Velocity at 45° Plane for $Re_{inlet}=40,000$	139
Figure 4.19 Measurements of Mean and rms Velocity at 90° Plane for $Re_{inlet}=40,000$	141
Figure 4.20 Measurements of Mean and rms Velocity at 135° Plane for $Re_{inlet}=40,000$	143
Figure 4.21 Measurements of Mean and rms Velocity at 180° Plane for $Re_{inlet}=40,000$	145
Figure 4.22 Predicted Streamwise Velocity Profile for $Re_{inlet}=15,000$	147
Figure 4.23 Predicted Radial Velocity Profile for $Re_{inlet}=15,000$	148
Figure 4.24 Comparison of Measured and Predicted Streamwise Velocity at 0° Plane for $Re_{inlet}=15,000$	149
Figure 4.25 Comparison of Measured and Predicted Radial Velocity at 0° Plane for $Re_{inlet}=15,000$	150
Figure 4.26 Comparison of Measured and Predicted Streamwise Velocity at 45° Plane for $Re_{inlet}=15,000$	151
Figure 4.27 Comparison of Measured and Predicted Radial Velocity at 45° Plane for $Re_{inlet}=15,000$	152
Figure 4.28 Comparison of Measured and Predicted Streamwise Velocity at 90° Plane for $Re_{inlet}=15,000$	153
Figure 4.29 Comparison of Measured and Predicted Streamwise Velocity at 135° Plane for $Re_{inlet}=15,000$	154
Figure 4.30 Comparison of Measured and Predicted Streamwise Velocity at 180° Plane for $Re_{inlet}=15,000$	155
Figure 4.31 Comparison of Measured and Predicted Streamwise Velocity at Exit Plane for $Re_{inlet}=15,000$	156

Chapter 5	
Figure 5.1	Flow Arrangement 190
Figure 5.2	Details of Vaporiser 191
Figure 5.3	Combustor Geometry 191
Figure 5.4	Finite Difference Mesh used in the Calculations 192
Figure 5.5	Measurement Grids for the Combustor 193
Figure 5.6	Mean Temperature Measurement at $x=36\text{mm}$ 194
Figure 5.7	Measured Temperature Distributions in Exit Plane for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=43$ 195
Figure 5.8	Distributions of Measured Species Concentrations in Exit Plane for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=43$ 196
Figure 5.9	Distributions of Calculated Equilibrium Concentrations in Exit Plane for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=43$ 197
Figure 5.10	Distributions of Measured Species Concentrations and Calculated Enthalpy Balance Temperature in Primary Zone for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=26$ 198
Figure 5.11	Distributions of Calculated Equilibrium Concentrations and Temperature in Primary Zone for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=26$ 199
Figure 5.12	Distributions of Measured Species Concentrations in Exit Plane for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=26$ 200
Figure 5.13	Distributions of Calculated Equilibrium Concentrations in Exit Plane for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=26$ 201
Figure 5.14	Calculated Equilibrium and Enthalpy Balance Temperature Distributions in Exit Plane for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=26$ 202
Figure 5.15	Streakline Plot of Isothermal Flow of Combustor 203
Figure 5.16	Streakline Plot of Isothermal Flow of Combustor in

Cross-Stream Planes	205
Figure 5.17 Predicted Passive Scalar Distributions in Isothermal Flow	206
Figure 5.18 The Influence of Dilution Jet Entry Angle to Jet Impingement Region in Isothermal Flow	207
Figure 5.19 Streakline Plot at $z=-42\text{mm}$ for Different Dilution Jet Entry Angle in Isothermal Flow	209
Figure 5.20 Streakline Plot at $z=-28\text{mm}$ for Different Dilution Jet Entry Angle in Isothermal Flow	211
Figure 5.21 Flow Visualisation of Combustor (Taken from Tse (1988))	213
Figure 5.22 Streakline Plot for Increased Mass Flow at Slot 3 in Isothermal Flow	214
Figure 5.23 Streakline Plot for Increased Mass Flow at Slot 4 in Isothermal Flow	216
Figure 5.24 Streakline Plot for Increased Mass Flow at Slot 3 and 4 in Isothermal Flow	218
Figure 5.25 Predicted Passive Scalar Distributions for Non-Uniform Velocity Profile at Vaporiser Exit in Isothermal Flow	220
Figure 5.26 Predicted Passive Scalar Distributions for Second Component Velocity of 15% in Isothermal Flow	221
Figure 5.27 Predicted Passive Scalar Distributions for Second Component Velocity of 25% in Isothermal Flow	222
Figure 5.28 Distributions of Thermochemical Properties for the Equilibrium and Laminar Flamelet Models	223
Figure 5.29 Percentage Difference in Density for the Equilibrium and Laminar Flamelet Models	225

Figure 5.30 Streakline Plot of Combusting Flow of Combustor for $T_{inlet}=315K$ and $AFR=29$	226
Figure 5.31 Streakline Plot of Combusting Flow of Combustor in Cross-Stream Planes for $T_{inlet}=315K$ and $AFR=29$	228
Figure 5.32 Predicted Species Concentrations Distributions with Equilibrium Model for $T_{inlet}=315K$ and $AFR=29$	229
Figure 5.33 Predicted Species Concentrations Distributions with Equilibrium Model for $T_{inlet}=315K$ and $AFR=29$ at $x=36mm$	232
Figure 5.34 Predicted Species Concentrations Distributions with Laminar Flamelet Model for $T_{inlet}=315K$ and $AFR=29$	233
Figure 5.35 Predicted Species Concentrations Distributions with Laminar Flamelet Model for $T_{inlet}=315K$ and $AFR=29$ at $x=36mm$	236
Figure 5.36 Streakline Plot of Combusting Flow of Combustor with Laminar Flamelet Model for $T_{inlet}=315K$ and $AFR=29$	237
Figure 5.37 Measured Temperature and Species Concentrations with Natural Gas as Fuel (Taken from Tse (1988))	239
Figure 5.38 Calculated Enthalpy Balance Temperature in Exit Plane for $T_{inlet}=515K$ and $AFR=43$ in Exit Plane	243
Figure 5.39 Comparison of Velocity Measurements and Prediction in Primary Zone	244
Figure 5.40 Comparison of Velocity Measurements and Prediction in Exit Plane at $y=0mm$	246
Figure 5.41 Comparison of Passive Scalar Measurements and Prediction at $y=0mm$ and $z=28mm$	247
Figure 5.42 Measured Species Concentrations Distributions	248

Figure 6.13 Distributions of Measured Major Species Concentrations in Exit Plane for $T_{inlet}=515K$ and AFR=60	277
Figure 6.14 Distributions of Measured Major Species Concentrations in Exit Plane for $T_{inlet}=515K$ and AFR=36	278
Figure 6.15 Comparison of Isothermal Streamwise Velocity With and Without the Curved Nozzle at $x=60mm$	279
Figure 6.16 Profiles of Isothermal Streamwise Velocity Without Curved Nozzle at $x=110mm$	281

NOMENCLATURE

Roman characters

A	cross-sectional area of the combustor
a	constant in β -function distribution
b	constant in β -function distribution
AFR	air-fuel ratio (mass basis)
CH ₄	methane
CO	carbon monoxide
CO ₂	carbon dioxide
C _p	specific heat capacity at constant pressure
C _v	specific heat capacity at constant volume
C _{μ} , C _{ϵ1} , C _{ϵ2}	constant in turbulence model
c	specific heat capacity of thermocouple
d	diameter of thermocouple
EI _{CO}	emission index of CO
EI _{UHC}	emission index of UHC
f _s	shift frequency
f _r	angular velocity of diffraction grating
g	gravitational force
h	local heat transfer coefficient
H ₂	hydrogen
H ₂ O	water vapour
J	Jacobian
k	turbulent kinetic energy
L	width of control volume
l	characteristic length scale in near wall region
LDV	laser-Doppler velocimetry

m	mass flow rate of combustor
m_{α}	mass of the species
N	size of the sample
N_2	nitrogen
P	pressure
pdf	probability density function
Pe	Peclet number
P_m	total pressure
P_s	static pressure
OTDF	pattern factor
O_2	oxygen
Re	Reynolds number
rms	root mean square
rpm	revolution per minute
S	source term in conservation equation
t	time
T	temperature
T_c	compensated temperature
T_{eq}	equilibrium temperature
T_g	temperature of the gases
T_h	enthalpy balance temperature
T_{inlet}	inlet temperature
T_m	measured temperature
T_{max}	Maximum measured temperature
T_{mean}	mean exit temperature
T_{th}	temperature measured by thermocouple
U	streamwise velocity
U_b	bulk mean streamwise velocity

U_c	bulk mean streamwise velocity in combustng flow
U^+	non-dimensional velocity in boundary layer
u	fluctuation of streamwise velocity
u_g	uncertainty of fluctuating component of velocity
V	normal velocity
v	fluctuation of normal velocity
W	tangential velocity
w	fluctuation of tangential velocity
y	distance from wall
y^+	non-dimensional distance from wall
Z_c	constant

Greek characters

α	quadratic function for curvilinear transformation
β	quadratic function for curvilinear transformation
δ_{ij}	kronecker's delta
ε	dissipation rate
Φ	scalar variable
Γ	diffusive coefficient
γ	quadratic function for curvilinear transformation
η	co-ordinates in transformed plane
κ	von Karman's constant
μ	viscosity of fluid
μ_t	turbulent viscosity of fluid
θ	angle of the curved duct
ρ	density of fluid
$\sigma_k, \sigma_\varepsilon$	constant in turbulent model
σ_t	turbulent Prandtl number

τ shear stress

Subscripts

1, 2, 3, port of pressure probe
4, 5
i, j, k co-ordinates
max maximum value
min minimum value
n number of node
res residual
s static
N, E, S, index for neighbouring cell of node P
W, L, R
U, P source term

Superscripts

- unweighted average
~ weighted average
' fluctuation of unweighted quantity
" fluctuation of weighted quantity

CHAPTER 1 INTRODUCTION

1.1 Introduction

The development of the gas turbine engine has been rapid for the past several decades and its application has been extended to marine and land-based power plants. More recently, the requirements of better efficiency and the limitations of noise and pollution, particularly oxides of nitrogen and carbon monoxide, have encouraged the need for better understanding of the complex fluid dynamical and chemical processes inside the engine.

Considerable research has been carried out to study the processes controlling the mixing and burning of the fuel inside the combustor and, as a consequence, the general nature of the flows and combustion processes are known as discussed, for example, by Lefebvre (1983) and Mellor (1990). There are three basic types of combustor employed in gas turbine engines: tubular, tubo-annular and annular as shown in Figure 1.1. The tubular type combustor is used mainly on centrifugal compressor engines and earlier types of axial flow compressor engines. It has the form of a cylindrical tube, or liner, mounted concentrically inside a cylindrical casing. The number of chambers varies from seven to sixteen situated circumferentially between compressor and turbine. Air is supplied from the compressor through each individual casing and into the inner flame tube in the form of jets and air-films for the purpose of combustion and wall cooling. Fuel is introduced through a vaporiser or an atomiser usually located on the combustor centreline in the head of the flame tube. The flame tubes are interconnected to allow each tube to operate at the

same pressure and combustion is then able to propagate around the flame tubes during ignition.

Most of the fuel is consumed in the upstream region of the flame tube, called the primary zone, with only about 40% of the air entering the chamber. The efficient combustion processes are achieved through a flame-stabilising recirculating flow in the primary zone. In tubular-type combustors, this is usually achieved by swirling air flow, creating a double-vortex recirculation, around the fine atomised spray of fuel. The air not used for combustion is introduced progressively into the downstream region of the flame-tube, the dilution zone, for wall-cooling and to lower the gas temperature from above 2000°C to a level at which the turbine blades can operate.

The main difference between the tubular and turbo-annular type combustor is that several flame tubes are fitted inside a common casing in the latter type, but the general nature of the flow is similar in both cases.

The annular combustor consists of a single flame tube in the form of an annulus contained in an inner and outer casings. It has the advantages of shorter length and less weight and the problem of combustion propagation from chamber to chamber is eliminated. In addition, a short shaft length is essential to overcome shaft whirling, as mentioned by Lefebvre (1983), and this has led to the use of reverse-flow annular combustors, Figure 1.2, which are widely used in small engines. In the reverse-flow annular combustor, the air that flows through the inner annulus suffers a higher pressure loss, due to its longer path, than the outer annulus and the flame cannot be easily stabilised by the double-vortex recirculating flow as in the tubular type combustor due to the difficulty in balancing the air jets from the inner and outer annulus in terms of jet entry angle, mass and momentum flow rate. Instead, a single vortex recirculation zone, through

a combination of single-sided air jets and film-cooling wall jets, is often used for flame stabilisation. The hot gas is then cooled with additional air entering further downstream before being directed to the inlet guide vane of the turbine module by the reverse-flow nozzle.

In the present study, the flow and combustion characteristics of a model of a reverse-flow annular combustor have been investigated. This Gem combustor, which is one of the first turboshaft engines to be wholly modular, has been the most important engine in the Small Engine Division in Rolls Royce, Leavesden for the past twenty years, as described by Gunston (1989), and more than 97% of the Gems have been used in helicopters. The fuel is supplied to the Gem combustor, Figure 1.3, through a J-shaped vaporiser and air through primary and dilution holes and film-cooling slots. In the model configuration adopted in the present investigation, a T-shaped vaporiser was used instead of the J-vaporiser in accord with current engine development, since the T-vaporiser has been adopted in the design of the latest small turboshaft engine, the RTM322.

The design of gas turbine combustors relies on empirical correlations, based on experimental data, which simplify the complex geometries and flowfields, often into steady one-dimensional flows as discussed by Mellor (1972, 1976). Although this approach has been relatively successful over many years, the continuous development of combustors has led to the need for detailed velocity, temperature and species concentrations measurements to ensure better understanding of combustor operation and improvements to calculation methods based on numerical solutions of differential equations which represent conservation of mass, momentum and energy. The latter can be used to provide useful information of the properties of combusting flows inside combustors but,

because of the need for assumptions, should be considered interpolative with extrapolation undertaken with great care.

The testing of new arrangements in model configurations is also essential for the successful development of combustors. Previous work of Tse (1988) provided velocity and scalar characteristics in a sector of a model Gem, representing only the primary and dilution zone and using natural gas as fuel. In continuing this work, the curved exit nozzle of the real engine was represented in the present model and kerosene was used as fuel to provide more realistic operating conditions. Numerical work was also carried out to demonstrate the extent to which geometric and flow variables can be represented. The flow through the curved exit nozzle has also been studied in detail with a separate and simplified configuration, both experimentally and numerically.

Previous work related to the present study is reviewed in the next section. Section 1.3 provides a brief description of the contribution of the present work and the last section in this chapter gives an outline of the remainder of this thesis.

1.2 Previous Work

This section provides a summary of previous work carried out in the field of gas turbine combustors. Section 1.2.1 reviews the measurements and predictions in curved bend configurations of relevance to the study of the flow through the curved exit nozzle. The next subsection concentrates on the measurements of velocity and scalar quantities inside combustion chambers. Although there are several types of combustion chamber, different in size, concept and method of fuel and air supply, the aerodynamic and combustion features are common to most of them. The

last subsection provides descriptions of calculations in combustor configurations and assesses their merits for representing flowfields.

1.2.1 Bend Flow

The curved exit nozzle found downstream of the dilution zone of the reverse-flow combustor is a 180° bend with an entry-to-exit area ratio of 5 and four cooling slots on the two curved surfaces with highly non-uniform inlet velocity and temperature distributions. In order to separate the influence of the curved geometry from factors such as the film-cooling slots within the bend, the complex arrangement was modified to a simple 180° bend with uniform inlet profiles in the study of a bend flow described in Chapter 4.

Investigations of flow through curved channel, listed in Table 1.1, have a wide range of practical applications including aircraft intakes, airfoil cascades in compressors and curved diffusers or nozzles. The cross-stream secondary motions in flows in ducts with curvature are induced by the transverse pressure gradient and diffusion in the Reynolds stresses in the cross-stream plane as reviewed by Johnston (1978). Nevertheless, the magnitude of the pressure driven secondary flow is an order of magnitude higher than the stress induced motions.

The secondary flow in a two-dimensional cascade of airfoils was studied by Squire and Winter (1951) providing quantitative measurements for comparison with inviscid flow theory. The results showed that the inviscid theory, which suggests that thinner entry boundary layers in the turbulent flow give rise to smaller cross stream velocities, gave a satisfactory approximation to the magnitude of the secondary flow in a cascade with a 90° corner and a large variation of approach velocity confirming the dominance of the pressure distribution on the flowfield.

The general nature of the flow through a 180° curved pipe has been discussed by Rowe (1970), by relating the local total pressure gradient to the production of streamwise vorticity. From the start of the bend, the secondary flow increases to a maximum at a bend angle of about 30° . From that point onwards, the total secondary flow is reduced until by about 90° , it reaches a steady value and after this, the flow can be considered to be fully developed with no further change in the flow pattern.

Laminar flow in a curved 180° square duct was studied, using laser-Doppler anemometry, by Hille et al (1985) with a relatively mild curvature of radius-to-width ratio of 6.45. The development of the streamwise velocity was found to be connected with a strong momentum transfer towards the outer wall between the entry plane and 60° plane, with a partial back transfer of the momentum towards the duct centre between 45° and 108° , and with little further change of the momentum between 108° and 180° near the outer wall. The maximum streamwise velocity, about 1.25 times that of the bulk mean velocity, remained close to the outer wall through the duct.

For a curved duct with sharp curvature, velocity measurements, using pressure probes, were made by Brown and Marris (1963) in a curved channels with constant cross-sectional area and with radius-to-width ratio of 1.83. An appreciable secondary flow, limited in the end plate regions, was recorded with a depth-to-width ratio of 8 whereas the maximum streamwise velocity, 1.2 times the bulk mean velocity, remained close to the inner wall, as a result of the strong radial pressure gradient, contrary to the case of laminar flow with high radius-to-width ratio. Similar flow development near the inlet region, with streamwise velocity profiles skewed towards the inner wall, was also observed by Taylor, Whitelaw and Yianneskis (1981) in a 90° bend with a radius-to-width ratio of 2.3. This

was due to the strong radial pressure gradient towards the inner wall, as a result of the sharp curvature and the reduced effect of the secondary flow in turbulent flows with thin inlet boundary layers. The results in the bend flow of Chapter 4, with a radius-to-width ratio of 1.2, a depth-to-width ratio of 3.8 and an entry-to-exit area ratio of 2.3, would then be able to quantify the extent of the influence of the secondary flow in the lateral direction and the effect of the area contraction of the curved exit nozzle where previous studies have concentrated on diffuser-type geometries such as those of Rojas, Whitelaw and Yianneskis (1983), for which in a 180° diffuser with mild curvature, the flow in the diffuser is qualitatively similar to that in constant area ducts.

In the reverse-flow nozzle found in the actual combustor, the two curved surfaces form an annulus passage with no solid boundary in the lateral direction, i.e. with an infinite aspect ratio. Therefore, the results in the curved bend, with rectangular cross-section, in Chapter 4 and the flow through the curved exit nozzle in the sector model of the combustor in Chapter 6, should be interpreted with great care for the discussion of flow in the actual nozzle. In a model combustor without a curved exit nozzle, similar to that employed in Chapter 5, Toral and Whitelaw (1982) confirmed that the end wall did not influence the mean flow characteristics in the major part of the combustor. Nevertheless, with curved geometries, the effects of aspect ratio have to be considered and its influence in the fully developed flows inside a 270° curved duct, with a radius-to-width ratio of 4, was studied with a combination of pressure probe and hot-wire anemometer, by Kita et al (1987). A range of aspect ratios between 0.4 to 16 was employed and it was suggested that in the case of low aspect ratios, the pressure driven secondary flow had a forced vortex type velocity profile in contrast to a free vortex type velocity distribution at high aspect

ratios. The results in Chapters 4 and 6 would provide quantitative answer to the effects of end wall for flow in curved geometries.

The inlet velocity profile is essential to the understanding of the flow development inside curved ducts, as discussed by Berger et al (1983), and detailed measurements, using laser-Doppler velocimetry, were made by Humphrey, Taylor and Whitelaw (1977), Humphrey, Whitelaw and Yee (1981) and Taylor, Whitelaw and Yianneskis (1981) in developing and fully-developed laminar and turbulent flow in a 90° square duct with strong curvature. It was observed that thicker initial boundary layer, as in the case of developing laminar flow, led to stronger secondary flow with velocity maximum of 60% of the bulk flow velocity compared with 40% in turbulent flow whereas the normalised velocity fields were similar in magnitude for fully-developed flows.

Numerical methods for predicting flow developments inside curved geometries can be divided mainly into two categories. The first is based on a partial-parabolic formulation of the governing equations, as described by Pratap (1975), with the assumption that recirculation in the plane parallel to the symmetry plane does not exist. The second approach is to solve the elliptic differential equations in finite difference form as described by Humphrey, Whitelaw and Yee (1981). Calculations of flow through a U-bend of square cross-section, based on a partial-parabolic formulation, were carried out by Chang et al (1983) with a two-equation eddy viscosity turbulence model and two differencing schemes, namely, HYBRID and QUICK and by Choi et al (1989) with an algebraic second-moment closure. The use of a higher order differencing scheme and second order turbulence model led to an appreciable improvement in the agreement between measurements and predictions, but the discrepancies between experimental data and computed results in the Reynolds stress field were still apparent.

The calculations by Humphrey, Whitelaw and Yee (1981) suggested that for pressure driven secondary flow, the calculated mean velocities were not strongly dependent on turbulence model and numerical errors are of greater importance for accurate solutions. Consideration of numerical uncertainties in the calculations of Chapter 4 is essential to the assessment of the ability of the numerical procedure to calculate flow in curved ducts. The problem of grid dependence is also considered in Chapter 4. Moreover, the measurements and calculations in Chapter 4 provide an alternative test case for assessing the extent to which flow variables can be represented with existing methodology.

1.2.2 Gas Turbine Combustor

Isothermal experiments in terms of water flow models, for example, by Green and Whitelaw (1980) and Koutmos (1985), have been used as simple and cheap means to provide information of the flow in gas turbine combustors. Although the inert flow can provide useful information of trends of major features resulting from changes in geometries and flow boundary conditions, important aspects of combusting flow cannot be reproduced in water models. Comparison between isothermal flow concentrations and combusting flow equivalence ratios, in the model annular combustor by Tse (1988), showed that they had similar features but with local differences in values of mean equivalence ratios of up to 90% in the lower half of the combustor, with high equivalence ratios, and of up to 40% in the remainder of the combustor. Hence, combusting flow results are indispensable to illustrate the more realistic flow patterns inside combustors under normal operating conditions and Table 1.2 gives a brief summary of experiments performed for this purpose.

The vaporisation of liquid fuel and fuel-air mixing processes in the primary zone are essential in determining the combustion efficiency of the combustor. Detailed measurements of species concentrations by Shisler et al (1975) in the Allison T-56 combustor and Vranos and Taback (1976) in the Pratt and Whitney JT 8D showed that film-cooling alone resulted in poor mixing in the primary zone and that reaction continued beyond the combustor exit. The method of air supply in the primary zone of the combustor in Chapter 5 provided an alternative arrangement which included a row of primary jets and additional film-cooling slots for the generation of a primary zone vortex and a T-vaporiser for fuel supply; its ability to vaporise and to mix the fuel with air are quantified by temperature and species concentrations measurements in the primary and dilution zone, as described in Chapter 5.

The importance of primary zone mixing was also illustrated in a can-type combustor between the model Spey combustor studied by Jones and Toral (1983), Bicen and Jones (1986) and Heitor and Whitelaw (1986) and the model Tay combustor of Bicen, Tse and Whitelaw (1990). The large swirler flow of the Spey combustor led to a streak of swirler air which passed down the centre of the combustor, preventing impingement of the primary jets and reducing their contribution to the primary zone as in the results obtained by Noyce et al (1981) and Noyce and Sheppard (1982). A reduction in the flow through the swirler from the 25% of total air flow in the Spey configuration to 7% in the Tay arrangement led to an equivalence ratio of 1.2 in the primary recirculation zone and primary jets which penetrated to the centre of the combustor, in contrast to a higher equivalence ratio of 2.0 in the primary zone in the former case. Hence, super-equilibrium CO and UHC did not occur in the Tay combustor and better combustion efficiency resulted.

Flow and combustion characteristics in can-type combustor have been studied, both in terms of simplified geometries, e.g. Altgeld et al (1983), Janjua et al (1983) and Rhode et al (1983), and more realistic configurations, e.g. LaRue et al (1984), Brum and Sameulsen (1987) and Cameron et al (1989) and a data base has been established to investigate the interaction and influence of different physical processes to combustor performance. The corresponding investigations concerning can-type and annular-type combustor were also carried out at Imperial College. The experiments of Jones and Toral (1983), Bicen and Jones (1986), Heitor and Whitelaw (1986), Bicen, Tse and Whitelaw (1990) and Tober (1990) have formed a comprehensive data base for can-type combustor. For annular-type combustor, Toral and Whitelaw (1982) studied the isothermal and combusting flows, with natural gas as fuel, of a simplified version of the annular combustor configuration investigated here. With a simple nozzle and gaseous fuel, the need for vaporisation of liquid fuel was removed. The measurements of velocity, temperature and species concentrations revealed the fuel-air mixing process in a single vortex primary zone recirculation which is also found in the model combustor of Chapter 5. Measurements of velocity and scalar fields were also made by Bicen, Tse and Whitelaw (1987, 1988) in a model combustor, identical to that in Chapter 5, for isothermal and combusting flows, with natural gas as fuel and without a reverse-flow nozzle. For both combustor configurations, equilibrium assumptions yielded concentrations and temperatures which were at variance with the measured values and by a considerable amount in fuel rich regions. The validity of the equilibrium assumption for liquid fuel can be examined through scalar measurements inside the model combustor in Chapter 5 for further development of the modelling of liquid fuel combustion inside gas turbine combustor.

Previous studies of a model annular combustor, by Toral and Whitelaw (1982) and Bicen, Tse and Whitelaw (1987, 1988), have concentration on simplified geometries or on part of the combustion chamber. The combustor in Chapter 6 represented the latest development in the study of the model annular combustor with geometry identical to the flame tube found in the actual configuration.

Preheated air was supplied to the combustors in Chapters 5 and 6 and the effects of preheat temperature have been studied in a can-type model combustor by Heitor and Whitelaw (1986) and in an annular-type model combustor by Bicen, Senda and Whitelaw (1988). With the inlet air temperature increased from 318K to 750K in a can-type combustor, the combustion efficiency increased from 84 to 99%, and resulted in an improved pattern factor from 0.70 to 0.60 based on 76% of the area of the exit plane. In the model annular combustor, the effect of increasing the inlet temperature by 200K was to increase the maximum and average exit temperature by about 35%, reduce the pattern factor from 0.55 to around 0.30 and reduce the emission indices of CO and UHC so as to increase the combustion efficiency from 69 to 94%.

The combustion of liquid fuel, used in the present study, is more complex than that of gaseous fuel since, in the former case, the evaporating process and initial droplet size produced by the atomiser influence the efficiency of combustion as reviewed by Lefebvre (1990). Liquid fuel combustion has been studied by Attya and Whitelaw (1981, 1984) in terms of a series of unconfined and confined kerosene spray flame and the influences of mean droplet diameter, swirl number and combustion air temperature were quantified. The flame with smaller initial droplets indicated a more rapid fall in droplet-number and density. The temperature and species concentration measurements made by Bhangu et al

(1983) in a combustor with kerosene as fuel provided validation of the design process but was still insufficient to quantify the extent of fuel breakdown in the combustor especially in the primary zone. Measurements of temperature and species concentrations in a model Tay combustor are in progress, as described by Tober (1990), to study the effects of using different fuels, when compared with the measurements of Bicen, Tse and Whitelaw (1990). The results in Chapter 5 provide measurements which help to quantify the effects of using different fuels in an annular combustor and hence to assess the validity of using gaseous fuel, instead of liquid fuel, in combustor research.

1.2.3 Numerical Modelling

The measurements discussed in the previous subsection provide data for the testing of calculation procedures used to simulate gas turbine combustor flowfields. The numerical approach of Chapters 4 and 5 is based upon the solution of the time-averaged Navier-Stokes equation and various aspects of the flow such as turbulence and combustion have to be modelled, as discussed by Jones and Whitelaw (1984). Table 1.3 provides a summary of numerical techniques and their application to the calculation of combustor flows.

A strong argument in favour of research in isothermal flows relevant to gas turbine geometries is that the predicted results can be compared with measurements for situations where the complications of combustion are absent. Thus, calculations of non-reacting flows provide a first step in numerical modelling of combustor flows. Green and Whitelaw (1983) simulated the flowfields inside a three-dimensional isothermal model combustor and compared with measurements to show discrepancies in the velocity field of up to 40% due, in part, to the incorrect

representation of the flow division in the annulus. Calculations of isothermal flowfields external and internal to the combustion chamber were carried out by Koutmos (1985) in a more realistic configuration and comparison of measurements and predictions in the annulus showed that the variation in the overall pressure recovery and loss coefficient could be predicted to within 7% of the inlet dynamic head. In the case of internal combustor flowfields, the predicted mean velocity distributions showed good agreement with measurements in the primary and intermediate zones, and the discrepancies were due to the poor performance of the turbulence model in regions of high anisotropy.

Prior to the calculations of combustor flows, it is desirable to determine the ability of turbulence models to deal with variable density flows. Pascau (1988) demonstrated that the second moment transport equation turbulence model was superior to the two-equation model in strongly swirling non-reacting flows but in the predictions of reacting flow in an axisymmetric combustor, there was no appreciable difference between results given by the two models. The merits of different numerical schemes have been studied by Alvarez (1990). A quadratic upstream differencing scheme, QUICK, was tested with a k - ϵ and Reynolds stress turbulence model and the problem of grid dependence was also addressed. The advantages of QUICK were obvious only in the regions of high level of turbulence where good agreement was observed between measurement and prediction in the normal stresses field.

Uncertainties in the inlet boundary conditions often lead to disagreement between measurements and predicted flowfields, for example, the isothermal and combustor flow calculations of Jones and McGuirk (1979) showed that inlet boundary conditions could lead to underestimation of recirculation lengths and temperature levels and that the two-equation

turbulence model also affected the accuracy of prediction in strongly swirling flows. The inlet boundary conditions have to be known before the predicted results can be interpreted correctly for design purpose. The parametric study of boundary conditions in Chapter 5 is useful for this purpose.

Multidimensional models for gas turbine combustor design, as described by Mongia (1987), have been developed to predict the combustion efficiency, CO, unburnt hydrocarbon, NO_x and soot emissions, pattern factor and wall temperature. Although the models cannot realistically predict detailed design parameters, they have been used to provide qualitative guidance in arriving at new design concept, Rizk and Mongia (1986).

A two-equation model and a fluctuating equilibrium scheme for combustion processes were applied to calculate the internal flowfields of one variant of the Rolls Royce RB 211 gas turbine combustor by Coupland and Priddin (1987) with a body-fitted orthogonal co-ordinates system which could be mapped into any axisymmetric internal flow region. The flowfields, exit temperature and NO_x emission were predicted to an accuracy acceptable for design purposes when compared with the limited experimental data. The calculations of three-dimensional flow in an annular combustor by Shyy et al (1989), with a computer program called CONCERT, as described by Burrus et al (1987), developed at GE Aircraft Engines, also indicated good agreement between experiments and predictions in the exit plane.

However, in the primary zone and with high equivalence ratios, the use of a equilibrium model to describe the thermochemical state of the mixture is inadequate, as suggested by Tse (1988). High concentrations of CO and H₂ resulted with the assumption of partial equilibrium state. A

better description of the combustion processes is necessary to account for the flow and combustion pattern in fuel rich regions. The laminar flamelet concept, as reviewed by Peters (1986), provides a useful alternative to model turbulent combustion and has been included for the combustor flow calculations of Chapter 5. Nevertheless, the assumption of fast chemistry is still needed and combustion was modelled to occur in asymptotically thin layers embedded within the turbulence flowfield. A comparison of equilibrium and one of the laminar flamelet chemistry models was made by McGuirk (1987) in a can-type combustor geometry and showed little change in the predicted mean density field and differences of less than 20% in mean velocity and mixture fraction distributions. Larger differences were observed in predicted gas temperature, up to 400K, and concentrations of CO and H₂ were significantly reduced in the fuel rich regions in the primary zone.

Liquid fuel complicates the prediction of combustor flows due to the need to model the two-phase flow problem of vaporisation and models for spray evaporation have been reviewed by Faeth (1979) and Mellor (1979). Calculations of kerosene-spray flames by El Banhawy and Whitelaw (1980) showed quantitative differences between measurements and predictions at near-burner locations and close to the combustion chamber centreline. The discrepancies were related to the uncertainties in the calculated spray evaporation and spreading rates. The number of droplet size ranges, representing the size-distribution within the spray, was found to be unimportant for the spray flame considered. The calculations of Attya and Whitelaw (1984) showed that droplet models were unnecessary with diameter of less than 50 μ m, and increasing with preheat temperature.

The locally homogeneous flow model of spray evaporation of Shearer et al (1979), which assumed infinitely fast interphase transport rates, was employed to predict the characteristics of an evaporating spray in stagnant air. The discrepancies in mean velocities and mixture fraction were less than 20% and 40% respectively and implied that the simplified spray model was insufficient to provide results with quantitative accuracy. These calculations suggested that models for liquid fuel combustion are still insufficient to provide quantitative agreement with measurements and further improvement is necessary to account for the turbulence diffusion of particles or droplets and the droplet evaporation.

The two-phase flow model of Gosman and Ioannides (1983) involved a solution procedure where the governing equations of each liquid phase droplet were solved by forward numerical integration, starting from the injection location at which the initial conditions were prescribed. In simple test cases where the initial conditions of the second phase could be prescribed exactly, good agreement with measurements were predicted but when the model was applied to a liquid-fuelled furnace where the initial conditions of the spray were estimated, significant discrepancies ^{were} observed between measurements and prediction. This highlights the importance of the spray initial conditions in calculations of combustor flow in complex configurations.

1.3 Present Contribution

The main purpose of the investigation reported in this thesis is to study the turbulent reacting flow in a reverse-flow annular combustor with a T-vaporiser for fuel supply, primary air jets to limit the primary zone,

film-cooling slots and dilution air jets for air supply and cooling and a reverse-flow exit nozzle.

Temperature and species concentrations were obtained in a model of the annular combustor without the reverse-flow nozzle to allow probe access to the primary and dilution zone. The combustor was operated at equivalence ratios corresponding to take-off and ground-idle conditions using kerosene as fuel. This provides knowledge of the combustion processes in upstream regions and, therefore, the difference in aerodynamic and combustion properties between the use of liquid and gaseous fuel can be quantified by comparison with previous measurements. Numerical simulations were also performed with the same geometry in isothermal and combusting conditions with gaseous fuel and the influence of variations in boundary conditions quantified. Two combustion models were tested to compare and contrast their merits for the calculation of combustor flows.

The measurements and predictions in the model of curved exit nozzle, with a simplified geometry and inlet conditions, provide a basis for analysing the flowfields in the actual nozzle. Velocity was measured to quantify the effect of the convergence and the magnitude of secondary flow within the flow domain.

Velocity, temperature and species concentrations measurements were obtained in a model of the annular combustor with the reverse-flow nozzle. Detailed velocity measurements inside the reverse-flow nozzle quantified the effects of its curvature, the presence of film-cooling slots within the nozzle and differences in non-reacting and combusting flows. The scalar characteristics at the exit of the combustor indicate the performance of the combustor at equivalence ratios corresponding to take-off and ground-idle conditions.

1.4 Layout of the Thesis

The remainder of this thesis is presented in six chapters.

Chapter 2 describes the experimental techniques including velocity measurements by pressure probe and laser-Doppler velocimetry, temperature by small diameter thermocouples and species concentrations by gas sampling probes. The uncertainties associated with the instrumentation are discussed.

Chapter 3 describes the calculation procedures. The exact equations are presented together with the turbulence and combustion models. Methods for discretising the partial differential equations, dealing with boundary conditions and mesh generation are discussed.

Chapter 4 presents the measurements and predictions of flow in a curved U-shaped nozzle. The discussion focuses on the development of flow structure and implications of the results for the application of the numerical scheme to this duct.

Chapter 5 deals with the experimental and numerical work carried out in a sector of a model annular combustor without the curved exit nozzle. The experimental results quantify the differences arising from the use of gaseous and liquid fuels and highlight the assumption of chemical equilibrium for combustion processes. The performance of the vaporiser is also assessed. The calculations are used to compare with previous experimental results and to explore the extent to which the flow and combustion characteristics can be represented by numerical methods.

Chapter 6 describes the velocity, temperature and species concentrations measurements obtained in a model annular combustor with a curved exit nozzle. The geometry is identical to that in the real combustor

but the method of air supply is different. The discussion examines the importance of the results to combustor design and the evaluation of combustor performance.

Chapter 7 summarises the main conclusions of the present investigation and provides suggestions for further research.

Table 1.1 Research in Bend Flow

Reference	Flow Configuration	Experimental Techniques or Numerical Procedures	Comment
Squire and Winter (1951)	2-D cascade of airfoils with non-uniform inlet velocity profiles	Pressure probe for longitudinal and radial velocity	Measurements show good agreement with results given by inviscid flow theory.
Brown and Morris (1963)	240° curved duct with depth-to-width ratio of 8 and mean radius-to-width ratio of 1.83	Dye trace flow visualisation; Pressure probe for longitudinal and radial velocity and static pressure	Appreciable secondary flow recorded due to the strong curvature and backflows observed at the inner wall.
Humphrey, Taylor and Whitelaw (1977)	Laminar flow in 90° square duct with radius ratio of 2.3 with fully-developed entry profile	Dye traces flow visualisation; Laser-Doppler velocimetry for longitudinal component of velocity; Fully elliptic form of Navier-Stokes equation was solved with standard tri-diagonal matrix algorithm	Calculated values exhibited the same trends as the experiments and in reasonable quantitative agreement.

Humphrey, Whitelaw and Yee (1981)	Turbulent flow in 90° square duct with radius ratio of 2.3 with fully-developed entry profile	Laser-Doppler velocimetry for longitudinal and radial components of mean velocity, and corresponding components of the Reynolds-stress tensor; Fully elliptic form of time-averaged Navier-Stokes equation was solved with an eddy-viscosity assumption	The secondary velocity, driven mainly by pressure field, were responsible for the convection of Reynolds stresses in cross-stream plane. The calculations were unable to represent the stress distribution within the flow.
Taylor, Whitelaw and Yianneskis (1981)	Developing laminar and turbulent flow in 90° bend with square cross-section and radius ratio of 2.3 and 7.0	Hydrogen bubbles for flow visualisation; Laser-Doppler velocimetry for longitudinal and radial components of velocity and associated Reynolds stresses; Pressure tapping for wall static pressure	Thicker boundary layer in the case of laminar flow leads to stronger secondary flow. The relative effect of secondary flow on the streamwise flow development is greater in the case of mild curvature.
Chang et al (1983)	Fully-developed turbulent flow in 180° bend with square cross-section and radius ratio of 3.35	Laser-Doppler velocimetry for longitudinal and radial components of velocity and associated Reynolds stresses; Finite difference method with two-equation eddy viscosity turbulence model with HYBRID and QUICK	Measurements are in qualitative agreement with results established from inviscid flow analysis but the numerical scheme failed to predict results quantitatively.

Hille et al (1985)	Fully-developed laminar flow in 180° square duct with radius ratio of 6.45	Laser-Doppler velocimetry for longitudinal and radial components of velocity	Strong streamwise momentum transfer towards the outer wall between 0° and 60°. Back transfer of momentum to the centre of the duct up to 108° and no further change of momentum downstream.
Azzola et al (1986)	Developing turbulent flow in 180° circular pipe with radius ratio of 3.375	Laser-Doppler velocimetry for mean and rms of longitudinal and radial components of velocity; Finite difference method with two-equation eddy viscosity turbulence model	Mean streamwise velocity changes little after 90° but the circumferential velocity never achieves a fully developed state Rate of recovery is underpredicted by the calculation.
Kita et al (1987)	Fully developed turbulent flow in a 270° curved duct with radius ratio of 4 and aspect ratios from 0.4 to 16	5-hole pressure probe and X-type hot-wire anemometer for mean and rms of longitudinal and radial components of velocity; Pressure tapping for wall static pressure	In the case of low aspect ratios, the main flow has a forced vortex type velocity distribution and has a free vortex type distribution at high aspect ratios.
Choi et al (1989)	Same as Chang et al (1983)	Finite difference method with algebraic second-moment closure turbulence model	Discrepancies between measurements and computed development are apparent although significant improvement resulted from the change of turbulence model and grid refinement in the viscous sublayer.

Table 1.2 Experiments in Gas Turbine Combustors

Reference	Experiment	Measurement Techniques	Comment
Shisler et al (1975)	Reacting flow in an Allison T-56 film-cooled combustor with liquid and gaseous propane fuel	Water-cooled probe for species concentrations	Little difference in NO exhaust emissions was observed as the fuel was changed from liquid to gaseous form.
Vranos and Taback (1976)	Reacting flow in a film-cooled Pratt and Whitney JT 8D combustor with liquid fuel	Steam-cooled probe for species concentrations	Quenching of the CO oxidation was evident in near wall region where the concentration of CO reached 10%.
Attya and Whitelaw (1981)	Unconfined Kerosene spray flame	Water-cooled probe for species concentrations; Suction pyrometer and 40, 80, and 300 μ m diameter Pt-Pt/13% Rh bare wire thermocouple for temperature	The flame with the smaller droplets developed faster with a higher centreline temperature than that with the larger droplets. Influence of suction rate and probe angle on concentration measurements was quantified.
Noyce et al (1981)	Reacting flow in a can-type research combustor with gaseous propane fuel	Water-cooled probe for species concentrations	The primary air addition method was shown to have a dominant influence on combustion

Noyce and Sheppard (1982)	Configuration same as Noyce et al (1981) but operated at a wide range of air-fuel ratios	Water-cooled probe for species concentrations	Substantial super-equilibrium CO levels persisted throughout the combustor indicating kinetic limitations for the combustion process.
Bhangu et al (1983)	Combusting flow in a tubo-annular combustor with Transply and kerosene fuel	Water-cooled probe for species concentrations; Bare wire thermocouple for temperature; Thermal paint and embedded thermocouple for wall temperature	The curved vane swirler, Transply and re-distribution of dilution holes led to reduction of CO, UHC and smoke.
Green and Whitelaw (1983)	Isothermal study of 3-D turbulent flow in an axisymmetric geometry with annulus fed dilution holes	Laser-Doppler velocimetry for mean and rms values of longitudinal velocity	The interaction between the annulus and the inner tube and the effect of primary jet trajectory on the upstream recirculation were quantified.
Jones and Toral (1983)	Isothermal and combusting flow in model can-type combustor, Spey, with porous wall, Transply, using gaseous propane fuel	Katharometer for passive scalar; Water-cooled probe for species concentrations; Suction pyrometer and 80 μ m diameter Pt-Pt/13% Rh bare wire thermocouple for temperature	Chemical equilibrium conditions prevailed only for locally fuel-lean conditions and CO to CO ₂ oxidation was partly controlled by finite rate chemical kinetics.

Toral and Whitelaw (1983)	Isothermal and combusting flow, with natural gas as fuel, in a sector of an annular combustor	Katharometer for passive scalar; Water-cooled probe for species concentrations; 80 μ m diameter Pt-Pt/13% Rh bare wire thermocouple for temperature; Laser-Doppler velocimetry and pressure probe for mean and rms values of the three components of velocity	Mixture fraction distributions, calculated from species concentrations, measured with combustion were similar to those of the passive scalar obtained in isothermal flow. Measured temperature exceeded calculated enthalpy balance temperature due to the differences between conventional and density-weight averages and the neglect of fluctuating temperature-concentrations correlations in the enthalpy balance.
Attya and Whitelaw (1984)	Confined Kerosene spray flame	Water-cooled probe for species concentrations; 80 μ m diameter Pt-Pt/13% Rh bare wire thermocouple for temperature; Embedded thermocouple for wall temperature; Laser-Doppler velocimetry for mean and rms values of velocity	The influences of spray mean droplet diameter, inlet air swirl and temperature were quantified.
Koutmos (1985)	Isothermal study of a can-type combustor with swirler, annular fed dilution holes and exit contraction nozzle	Laser-Doppler velocimetry for three components of mean and rms velocity both inside and outside the annulus	The interaction between swirler and primary jets and the effect of swirler on jet penetration, regions of recirculation and the generation of turbulence were quantified.

Bicen and Jones (1986)	Same as Jones and Toral (1983)	Laser-Doppler velocimetry for mean and rms values of the axial and swirl velocity components	Effect of air-fuel ratio on velocity characteristics is relatively weak in the primary zone but became steadily more pronounced with downstream distance.
Heitor and Whitelaw (1986)	Combustor is the same as Jones and Toral (1983) but with higher preheat temperature and different air-fuel ratios for combusting flow	Katharometer for passive scalar; Water-cooled probe for species concentrations; 80µm diameter Pt-Pt/13% Rh bare wire thermocouple for mean and rms fluctuating values of temperature; Laser-Doppler velocimetry for mean and rms values of the axial and swirl velocity components	Combustion increased the strength and decreased the width of the primary vortex. In the recirculation zone, gradient transport is negligible.
Bicen, Tse and Whitelaw (1987)	Isothermal flow in a model annular combustor, Gem	Flow visualisation using titanium dioxide; Katharometer for passive scalar; Laser-Doppler velocimetry and pressure probe for mean and rms values of the three components of velocity	The primary zone is characterised by a vortex driven by the cooling slots flow and primary jets. The asymmetric vaporiser exit profiles had a significant effect on the distribution of passive scalar.

Bicen, Tse and Whitelaw (1988)	Combusting flow in a model annular combustor, Gem, with natural gas as fuel	Water-cooled probe and quartz microprobe for species concentrations; 80 μ m diameter Pt-Pt/13% Rh bare wire thermocouple for mean and rms fluctuation values of temperature; Laser-Doppler velocimetry for mean and rms values of two components of velocity	The effect of air-fuel ratio was found to be small in the primary zone and modest in the exit plane. The exit plane pattern factors were higher than those of engine practice due to the atmospheric conditions.
Bicen, Senda and Whitelaw (1988)	Combusting flow in a model annular combustor, Gem with gaseous fuel and a 200K preheat	Water-cooled probe and quartz microprobe for species concentrations; 80 μ m diameter Pt-Pt/13% Rh bare wire thermocouple for mean and rms fluctuation values of temperature;	The effect of increasing the inlet air temperature was to increase the maximum and average exit temperature, reduce the pattern factor and emission index of CO and UHC so that the combustion efficiency increased.
Cameron (1988)	Isothermal and combusting in a axisymmetric combustor with liquid fuel	Laser-Doppler velocimetry for mean and rms values of velocity; Fine-wire thermocouple for temperature; Phase Doppler Interferometer for droplet size and velocity	The dome region recirculation was relatively insensitive to a change in dome geometry. Primary air provided closure to the dome and enhanced turbulence mixing.

Bicen, Tse and Whitelaw (1990)	Combusting flow in model can-type combustor, Tay, with gaseous propane fuel	Water-cooled probe for species concentrations; 80μm diameter Pt-Pt/13% Rh bare wire thermocouple for temperature	The effect of air-fuel ratio, preheat temperature were quantified. A reduction in swirler increased the penetration of the primary jets and combustion efficiency.
-----------------------------------	---	---	--

Table 1.3 Numerical Modelling of Gas Turbine Combustors

Reference	Configuration	Numerical Procedure	Comment
Jones and McGuirk (1979)	Isothermal and reacting flow in 2-D axisymmetric combustor with gaseous fuel and reacting flow in 3-D combustor with liquid fuel	Finite difference method for the time-averaged Navier-Stokes equation with a two-equation eddy viscosity turbulence model; Assumption of fast chemistry and turbulence/chemical interaction modelled via a β -function PDF; Fuel spray represented by transport equation for the PDF of droplet mass fraction and diameter	Lower level of mixing between fuel and air was predicted. Lack of knowledge concerning inlet boundary conditions were the main source of error.
Serag-Eldin and Spalding (1979)	Reacting flow in 3-D can-type combustor with swirler	Finite difference method for the time-averaged Navier-Stokes equation with a two-equation eddy viscosity turbulence model; Additional equations for total enthalpy, concentration of fuel and its fluctuation were solved	Limited comparison between measurements and predictions showed correct prediction of trends and quantitative disagreement was attributed to the combustion model

Shearer et al (1979)	Variable density jets and evaporating spray in stagnant air	Finite difference method for the time-averaged Navier-Stokes equation and a second-order turbulence model; Assumption of infinitely fast interphase transport rate	Prediction was in good agreement in single-phase flow. For two-phase flow, the discrepancies were more apparent due to the neglect of finite rate interphase transport.
El Banhawy and Whitelaw (1980)	Spray flame in simplified model of combustion chamber	Eulerian conservation equations for the gas phase and Lagrangian equations of droplet motion and thermal balance to a finite number of size ranges	Comparison between measurements and prediction revealed the importance of spray initial condition. The number of size ranges was found not to be important for the spray considered.
Gosman and Ioannides (1983)	Dispersion of single particles; Volatile and combusting sprays in simple solid-cone axisymmetry spray and hollow-cone swirling kerosene spray in a cylindrical combustion chamber	Eulerian conservation equations for the gas phase and "discrete droplet" model for the liquid phase	Application to simple test case produced reasonable agreement. For complex geometry, uncertainties were due to the inadequate knowledge of the initial size distribution, mean velocities and other properties of the droplets.

Green and Whitelaw (1983)	Isothermal study of 3-D flow in a simplified model can-type combustor	Finite difference method for the time-averaged Navier-Stokes equation with a two-equation eddy viscosity turbulence model	The discrepancies between measured and calculated results depended more on the distribution of grid than on the turbulence model.
Attya and Whitelaw (1984)	Confined kerosene spray flame	Same as El Banhawy and Whitelaw (1980)	Magnitude of associated uncertainties were quantified. For droplet size of less than 50 μ m, and increasing with preheat temperature, droplet models were unnecessary.
Koutmos (1985)	Isothermal study of a can-type combustor with swirler, annular fed dilution holes and exit contraction nozzle	Finite difference method for the time-averaged Navier-Stokes equation with a two-equation eddy viscosity turbulence model; 2-D boundary-fitted mesh for prediction of flow in the outer annulus	The poor performance of the turbulence model in regions of high anisotropy led to underestimated of turbulence energy. Detailed aspect of isothermal flow were predicted with accuracy adequate for preliminary purpose.

Coupland and Priddin (1987)	Reacting flow in RB211 annular combustor	Body-fitted orthogonal co-ordinate system; Finite difference method for the time-averaged Navier-Stokes equation with two-equation eddy-viscosity model; Combustion chemistry represented by a fluctuating scheme with assumption of chemical equilibrium	A smaller value of turbulent Prandtl number is used to correctly predict the lateral spread of fuel. The method used was adequate for engineering calculations of practical gas turbine combustion system.
McGuirk (1987)	3-D reacting flow in a can-type combustor with gaseous fuel	Finite difference method for the time-averaged Navier-Stokes equation with a two-equation eddy viscosity turbulence model; Either a full equilibrium or laminar flamelet description of instantaneous thermochemistry was used with a β -function PDF modelling the turbulence/chemistry interaction	The predicted flow patterns were the same for both chemistry model although difference of almost 100% were observed for the instantaneous density field. The levels of CO and H ₂ were significantly reduced with flamelet model.
Pascau (1988)	Variable density flow in an axisymmetric combustor	Finite difference method for the time-averaged density-weighted Navier-Stokes equation with a second order turbulence model and transport equations for scalar flux	For strongly swirling flow, the second order model was superior to the two-equation eddy viscosity model.

Shyy et al (1989)	Reacting flow in GE/SNECMA CFM 56 and GE CF6-80C annular combustor	Non-orthogonal curvilinear co- ordinates system; Finite difference method for the time-averaged Navier-Stokes equation with a two-equation eddy viscosity turbulence model; Combustion modelled through an assumed PDF and the conserved scalar variable formulation	For the normal operating condition where the physical process is likely to dominant, the performance of a gas turbine combustor can be predicted by the present methodology.
Alvarez (1990)	Jet in a crossflow	Finite difference method for the time-averaged Navier-Stokes equation with either a two-equation eddy viscosity or Reynolds stress turbulence model with HYBRID or QUICK differencing scheme with grid refinement	No grid independent solution was obtained. QUICK gave superior solution in regions of high turbulence level.

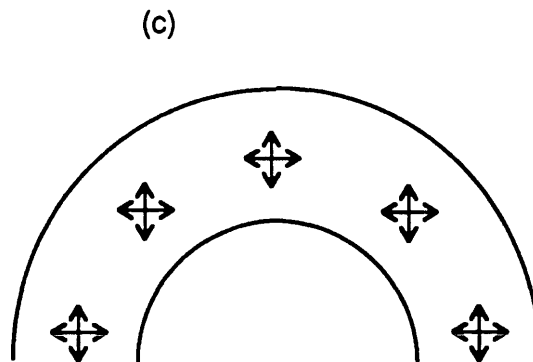
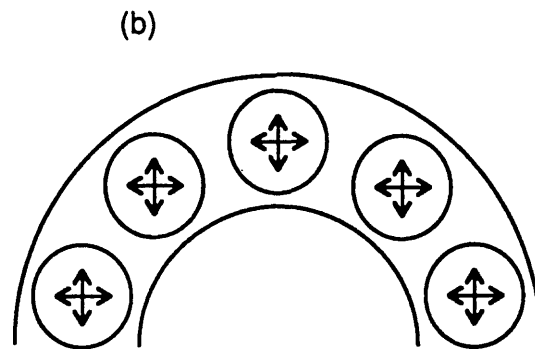
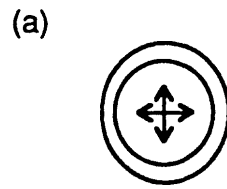


Figure 1.1 Three Types of Combustor
(a) Tubular
(b) Turbo-annular
(c) Annular

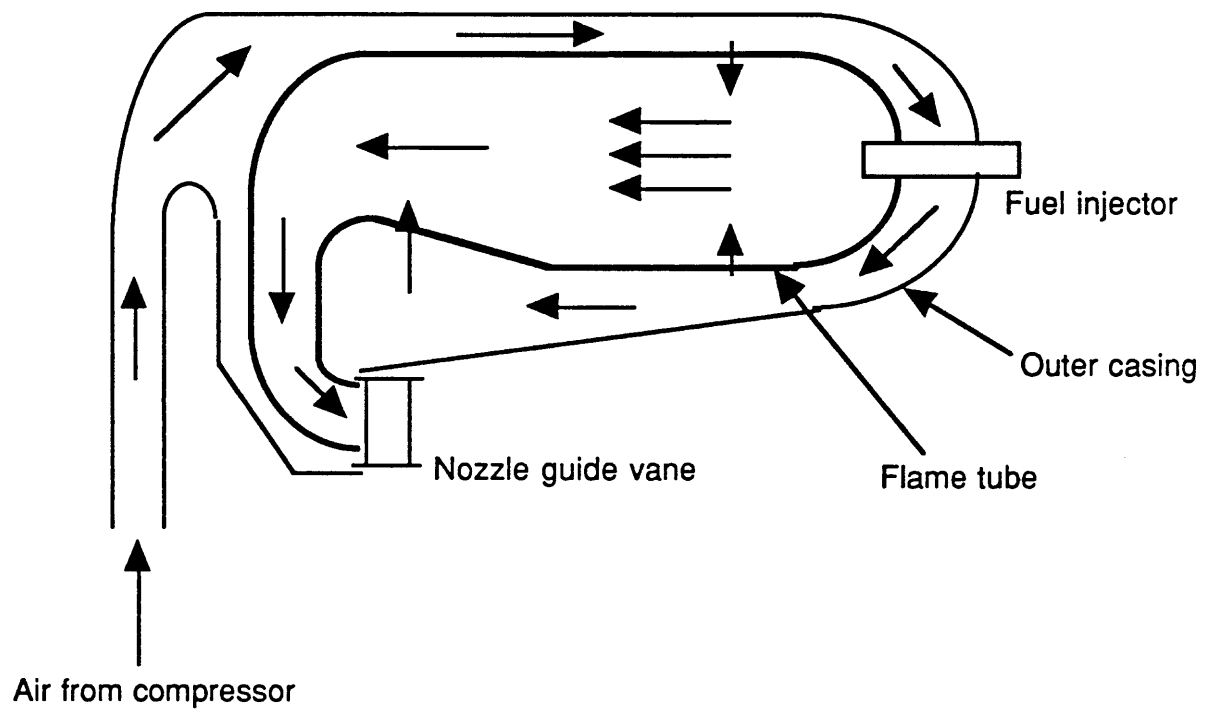


Figure 1.2 Reverse-Flow Annular Combustor

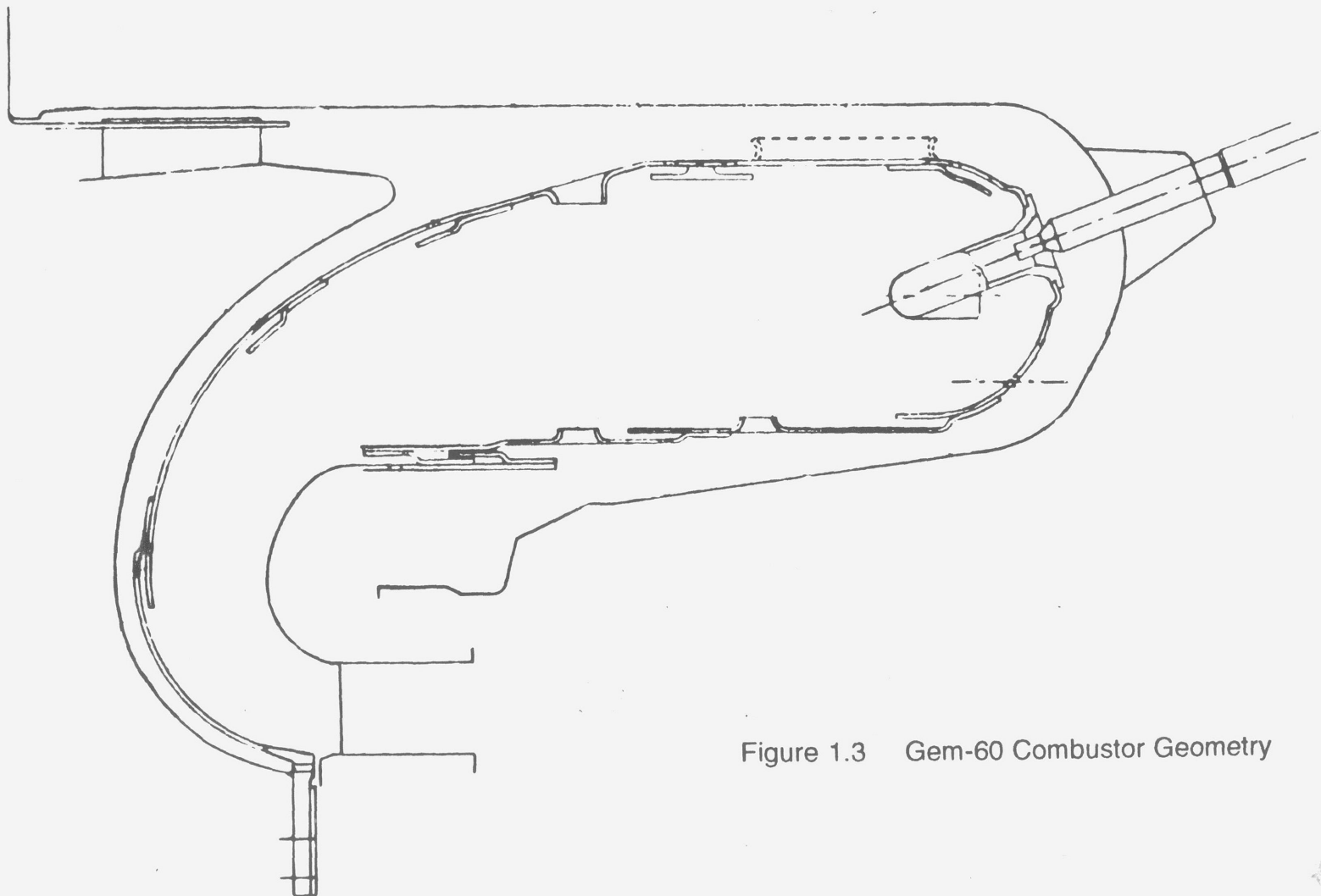


Figure 1.3 Gem-60 Combustor Geometry

CHAPTER 2 EXPERIMENTAL TECHNIQUES

2.1 Introduction

The study of the flowfields of gas turbine combustors requires detailed measurements of velocity and scalar characteristics inside the combustion chamber under isothermal and combusting conditions. This chapter introduces the techniques employed to measure velocity, temperature and species concentrations in the configurations investigated in the present study.

Probe techniques have been widely used in research and industry to measure velocity, temperature and compositions in combusting flows. However, the presence of a probe in the combusting flowfields can alter the local fluid dynamic, thermal and chemical properties so that the results are not always correct. Optical techniques have been developed to overcome this problem but require optical access and, at least for the measurement of scalar quantities, are less well established.

The well established techniques for velocity measurements include pressure probes, hot-wire anemometer and laser-Doppler velocimetry (LDV). Pressure probes can measure the mean velocity but not the fluctuating quantities and, because of its ease of use and simplicity of concept, the pressure probe is most suitable for testing and commissioning of new experimental arrangements and preliminary measurements. Although the hot-wire anemometer can measure the mean and fluctuating characteristics of velocity, the high temperature in combusting flows precludes its use and LDV was used in the present study because of its

robustness in terms of operating conditions and the absence of probe interference effects.

Measurements of temperature with optical techniques, although less well developed than for velocity, have been achieved through the use of, for example, Rayleigh scattering by Chandran et al (1984), Raman spectroscopy by Masri and Bilger (1988), infrared spectrometry by Zhang and Cheng (1986) and coherent anti-stokes Raman spectroscopy, CARS, by Magre et al (1988). However, their expenses and the need for experienced operators precluded their use in the present investigation and small diameter thermocouples, which offer the best compromise between cost, accuracy and ease of use, were preferred.

The measurements of species concentrations in flames with optical techniques, for example, Raman scattering, Rayleigh scattering and CARS, offer the advantages of not interfering with the combusting flow pattern and their insensitivity to hostile environments. But their development has not reached a stage where their usage is justified in terms of cost and accuracy and gas sampling analysis with sampling probes is still widely used in gas turbine industry. Hence, the use of sampling probes with gas analysers for major species was adopted in the present study.

The next section describes the techniques and their accuracy for velocity measurements with subsections dealing with pressure probes and LDV respectively. Section 2.3 deals with the use of small diameter bare-wire thermocouples for measurements of mean and rms temperature and the corresponding uncertainties. The instrumentation, sampling procedures and accuracy of species concentrations measurements are described in Section 2.4

2.2 Velocity Measurements

2.2.1 Pressure Probe

The measurements of symmetry and preliminary results in the bend flow of Chapter 4 were obtained with the 5-hole pressure probe shown in Figure 2.1, manufactured from hypodermic tubing with internal and external dimensions of 0.6 and 0.9mm respectively, and a micro-manometer with a precision of ± 0.01 mm of water. The probe was secured to a mechanism which allowed it to rotate in a vertical plane and was aligned to the local flow direction to give p_2 equals to p_4 while the central hole registered the total pressure value, p_5 . The relation between the resultant velocity, V_{res} , and the values of the pressure in each of the five holes is given by,

$$P_M = \frac{P_1 + P_2 + P_3 + P_4}{4} \quad (2.1)$$

$$V_{res} = \left(\frac{2}{\rho_{air}} \left(\frac{9.712368 (P_5 - P_M)}{0.64} \right) \right)^{0.5} \quad (2.2)$$

according to the calibration by Milosavljevic (1989) and a relation was also established between the pitch angle to the five pressure readings for velocities below 13m/s. With the yaw angle given by the orientation gear, the three components of velocity can then be resolved. However, due to the relatively wide separation of the side tubes for measuring p_2 to p_5 , about 1.5mm, the accuracy would be reduced in the presence of a steep gradient of pressure as mentioned by Bryer and Pankhurst (1971).

Nevertheless, the description of the flow by the incompressible Bernoulli equation,

$$P = P_s + 0.5 \rho U^2 \quad (2.3)$$

upon which the pressure probe technique is based, leads to uncertainties of less than $\pm 1\%$ for velocities less than 60m/s.

2.2.2 Laser-Doppler Velocimetry

Laser-Doppler velocimetry was used to measure the mean and rms values of two components of velocity in the curved bend in Chapter 4 and inside the model combustor in Chapter 6 in isothermal and combusting conditions. The velocimetry system is shown in Figure 2.2 with the optical specification of Table 2.1. The basic principles of LDV have been discussed in detail by Drain (1980) and Durst, Melling and Whitelaw (1981). The transmitting unit consisted of a 15 mW He-Ne laser and an integrated optical unit with a rotating grating module and two lenses. A lens was mounted on the laser head to focus the laser beam onto the grating disc and thereby to ensure that the two beams, split by the grating, were of circular cross section. The two lenses inside the optical unit focused the two beams to intersect at the location of measurement and form an ellipsoidal measuring volume. An imaging lens was then used to collect the forward scattered light, as a result of scattering particles entering the measuring volume, and focused it onto the pinhole of the photomultiplier. The whole assembly was secured to an optical bench and then to a milling table which provided traverse in three directions with a positioning accuracy of $\pm 0.1\text{mm}$. The signal from the photomultiplier was amplified, band pass filtered, and digitised by a frequency counter (DISA type

55L90a) which also included the power supply to the photomultiplier. The control of the counter and data processing was achieved through an Apple IIe microcomputer to provide the mean and rms velocity components.

Two seeding methods were used in the present study. In the investigation of the curved bend in Chapter 4, silicone oil droplets were added to the air flow immediately downstream of the diffuser through an atomiser which was simple and easy to use. In studying the velocity characteristics inside the model combustor during combustion, solid Kaolin particles were used, with equivalent aerodynamic diameter of $2\mu\text{m}$, and were dispersed in a spark generator and carried by compressed air to the upstream end of the combustor.

In flows with regions of recirculations, the provision of the direction of flow was necessary and the frequency corresponding to zero velocity was shifted so that the minimum velocity was represented by a positive frequency. The frequency shift was achieved through the rotation of the diffraction grating inside the optical unit and the relationship between the speed of rotation of the grating and the frequency shift is given by,

$$f_s = 2 N f_r \quad (2.4)$$

where f_s is in Hz, N is the number of lines in the grating and f_r is the angular velocity.

The uncertainties associated with velocity measurements made in a similar model combustor configuration have been discussed by Tse (1988) and only a brief description will be given in the following paragraphs. The main sources of uncertainty stemmed from the finite sampling statistics and velocity gradient broadening effects. The finite size of the control volume leads to the distortion of the probability density function of the Doppler

frequency and the contribution of a mean velocity gradient to the uncertainty of the fluctuation component of velocity is shown to be,

$$\sqrt{u_g^2} = \frac{L}{4} \frac{d\bar{U}}{dz} \quad (2.5)$$

according to Durst, Melling and Whitelaw (1981) where L is the length of the control volume and for the configurations investigated in the present study, the broadening effect can lead to uncertainties of less than 8% in the rms values.

The statistical uncertainties in the mean and fluctuation velocity are, according to Yanta and Smith (1978), given by,

$$\bar{U}_c = \frac{Z_c}{\sqrt{N}} \frac{\sqrt{u^2}}{\bar{U}} \quad (2.6)$$

$$\sqrt{u_c^2} = \frac{Z_c}{\sqrt{2N}} \quad (2.7)$$

where for a confidence limit of 95%, for which Z_c is equal to 2.0, and a minimum sample size of 2000, as used here, both uncertainties are less than $\pm 2\%$ for a typical value of turbulence intensity of 30%.

According to Glass and Bilger (1978), with the assumption of insignificant seeding and velocity bias effects, the measured velocity averages should be close to density weighted averages, as defined in Equation (3.7), and not to the normal time-averaged values as stated from Equation (3.4) to (3.6).

2.3 Temperature Measurements

The thermocouple probe and the associated data-processing system, for the measurements of mean and fluctuating values of temperature in Chapters 5 and 6, are shown in Figures 2.3 and 2.4 respectively. The thermocouple junctions were manufactured by butt-welding 80µm platinum and platinum/13% rhodium wires. The thermocouple was supported by 500µm wires of the same materials with a separation of between 10 to 15mm. The 500µm wires were embedded in a straight length of twin-bore ceramic tube of 2mm diameter which was placed inside a stainless steel tube of 4mm in diameter. The output of the thermocouple was amplified by about 100 and then digitised and processed by an Apple IIe microcomputer to provide the mean and fluctuating components of the temperature.

Due to the thermal inertia of the thermocouple, the relation between the temperature measured by the thermocouple and the actual gas temperature, according to Ballantyne and Moss (1977), is given by,

$$\frac{dT_{th}}{dt} = \tau^{-1} (T_g - T_{th}) \quad (2.8)$$

assuming that convective heat transfer to the thermocouple wire dominated radiative and conductive contributions. This requires the knowledge of the time constant of the thermocouple, τ , which is modelled as,

$$\tau = \frac{\rho dc}{4h} \quad (2.9)$$

for which ρ , d and c are density, diameter and specific heat capacity of the wire respectively. The the local convective heat transfer coefficient, h , is a

function of the local properties of the gas medium and flowfield. Various methods have been proposed to determine the time constant, for example, the pulse heating circuit of Lockwood and Moneib (1981). Heitor (1985) employed a similar method to establish a relation between the time constant and local gas temperature and concluded that, for the temperature range from 300 to 2000K, a linear variation from 17 to 6ms was sufficient to represent the fluctuating time constant for a 80 μ m thermocouple wire and this relationship was adopted in evaluating the rms temperature fluctuating in the present study. The compensated temperature can then be evaluated from the linearised form of Equation (2.8),

$$T_c = T_{th}^t + \tau \frac{T_{th}^{t+1} - T_{th}^t}{\delta t} \quad (2.10)$$

where δt depends on the sampling rate.

The uncertainties in the temperature measurements depend upon radiation from the wire, conduction to the main lead and catalytic effects. Since all the temperature measurements were made at the exit of the combustors in Chapters 5 and 6, the effects of flame stabilisation and probe interference were not likely to be important. The conduction loss was estimated using the method suggested by Bradley and Matthews (1968) and found to be less than 0.1%. Radiation loss, which is proportional to the local temperature, caused an underestimation of 5% at 1400K as suggested by Holderness et al (1969) and maximum uncertainties of $\pm 10\%$ at 1800K was reported by Heitor (1985). The catalytic effect can be important in regions of high concentration of unburnt hydrocarbon and would cause a maximum increase of measured mean temperature of up to 70K as suggested by Bicen, Heitor and Whitelaw (1986). Due to the low

concentration of unburnt hydrocarbon, less than 0.05% in mole fraction, measured at the exit of the combustors in Chapters 5 and 6, the catalytic effect of the bare-wire thermocouples is therefore unimportant in the present study.

One of the uncertainties caused by neglecting fluctuations in time constant was identified by Chomiak and Niedzialek (1967) who reported a possible displacement of measured mean temperature relative to the true temperature. However, in general, the time constant is unimportant in the measured mean temperature as suggested by Ballantyne and Moss (1977) and LaRue et al (1984). Instead, the uncertainties in the time constant can lead to a maximum discrepancy of $\pm 20\%$ in the rms values of temperature fluctuations for a $80\mu\text{m}$ thermocouple according to Heitor (1985). The comparatively low gas velocity imply that the thermocouple outputs were related to the local static temperature and the results were close to unweighted mean values as suggested by Toral and Whitelaw (1982).

2.4 Species Concentrations Measurements

The measurements of concentrations of CO , CO_2 , O_2 , H_2 , and unburnt hydrocarbon (UHC) in the model combustors of Chapters 5 and 6 were achieved by extracting gas samples through sampling probes to various analysers. Two probes have been used throughout the experiments: a stainless steel water-cooled probe and a quartz microprobe. The outside and inside diameter of the water-cooled probe, Figure 2.5, and of the quartz microprobe, Figure 2.6, were 8 & 1mm and 2.5 & 0.5mm respectively.

The sample was sucked through the probe by a pump and passed through a separator and filter to remove water and UHC. A vacuum pump maintained a back pressure to ensure shorter residence time and sufficient

quenching with the quartz microprobe. Otherwise, conversion of CO to CO₂ could occur in the uncooled probe as suggested by Schoenung and Hanson (1981). The performance of the two sampling probes has been studied and compared by Tse (1988) and the distribution of various species obtained with the two probes were similar and the larger dimension of the water-cooled probe resulted in some spatial averaging and quenching of the reaction. Although supersonic flow was not achieved inside the quartz microprobe, the quenching of the quartz microprobe was sufficient as in the results of Colket et al (1982).

The sampling line between the UHC analyser and the probe was electrically heated to about 150°C to avoid condensation of water and UHC so that the measured concentrations of CO, CO₂, O₂, H₂ were based on dry analysis whereas the concentration of UHC was based on wet analysis. The water-cooled probe was used to measure the composition at the exit to provide iso-kinetic sampling and the quartz microprobe was used inside the model combustor to minimise probe interference effects. The sampling systems are shown in Figures 2.7 and 2.8 for the quartz microprobe and the water-cooled probe respectively. The analysers were:

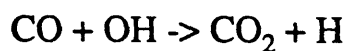
- (1) Infrared analyser for CO by Analytical Development Company;
- (2) Infrared analyser for CO₂ by Analytical Development Company;
- (3) Beckman E2 paramagnetic analyser for O₂;
- (4) Pye 'Series 104' chromatograph with heat conductivity detector for H₂;
- (5) Analysis Automation 520 Series II flame ionisation detector for UHC;

and were also used by Tse (1988) with the details of calibration and sampling procedures described by Godoy (1982) and Wilhelmi (1984).

There are two main factors that can influence the accuracy of concentration measurements: the first is probe interference and the second is associated with sample handling and analysis. Uncertainties in the measurement of the various gas concentrations were less than $\pm 0.5\%$ of the full scale of each analyser except for the gas chromatograph for which the accuracy is within $\pm 2\%$ of the full scale, but greater errors can result from the sampling process.

The suction velocity of the water-cooled probe was kept constant at around 20m/s to achieve near iso-kinetic sampling as mentioned by Bilger (1977) who has also shown that high suction velocities may cause a bias towards the lower density components. The suction velocities were within a range which, according to Heitor and Whitelaw (1986), implied that sampled gas concentrations were independent of the suction velocity. The suction velocity for the quartz microprobe was close to 100m/s and, according to Attya and Whitelaw (1981), implied a bias towards lower density components with values of CO and CO₂ which were high by up to 15%. The error due to the difference between the angle of the probe to the local flow direction was also quantified by Attya and Whitelaw (1981) whose results showed that the concentrations of CO and CO₂, obtained with an inclined probe, were lower by 25% and 3% respectively compared with those obtained with the probe axis parallel to the flame axis. In the present study, the orientation of the sampling probe was adjusted at the exit of the combustor so as to align to the direction of the bulk flow as indicated by the velocity measurements. Therefore, the error due to the mismatch between the angle of the probe and the local flow direction was negligible.

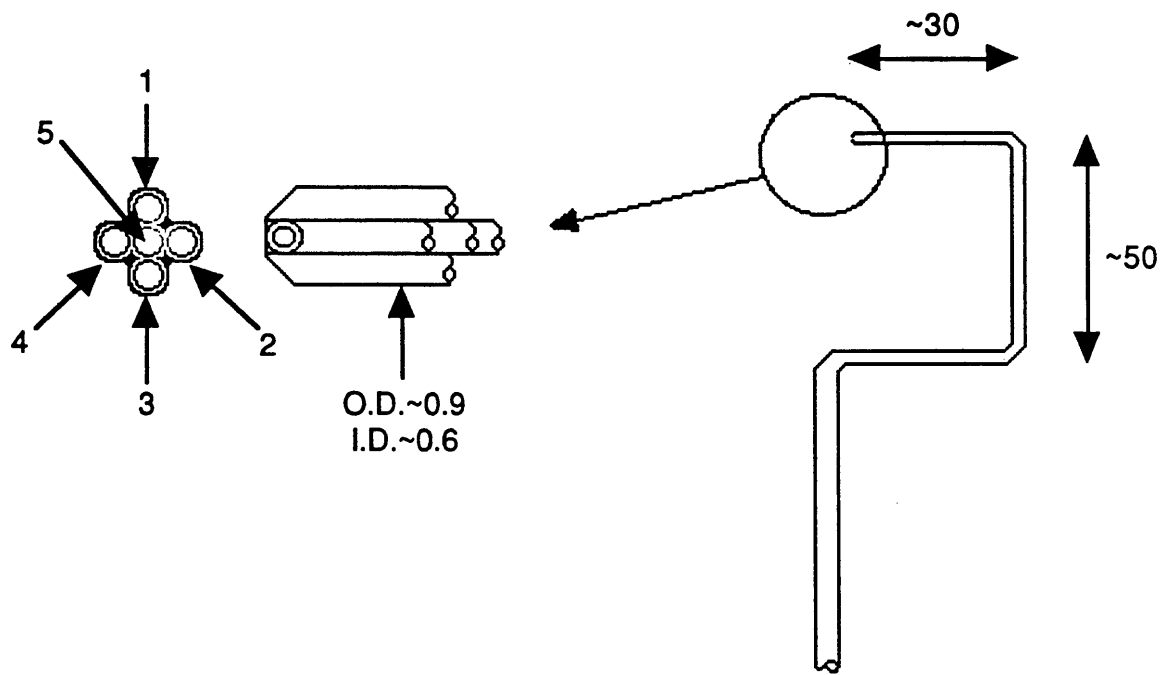
The loss of CO due to attack by active species through the reaction,



as discussed by Malte and Kramlich (1980) and in the presence of metal surface investigated by Kramlich and Malte (1980) and Shelef et al(1968) could not be readily quantified in the present experiment. The disturbance due to the presence of the probes to the flowfield was also important but no attempt had been made to quantified its effect. Nevertheless, from similar experiments of Toral and Whitelaw (1982), Attya (1983) and Heitor and Whitelaw (1986), the uncertainty of each gas concentration should be less than $\pm 10\%$ of the maximum concentration. The measured concentrations of major species are expected to close to density-weighted averages as suggested by Kennedy and Kent (1981).

Table 2.1 Characteristics of the Optical System

Wavelength of He-Ne laser (nm)	632.8
Focal length of focusing lens (mm)	80
Focal length of collimating lens (mm)	300
Focal length of imaging lens (mm)	300
Focal length of objective lens (mm)	100
Number of line pairs in diffracting grating	16384
Maximum frequency shift (MHz)	8.78
Fringe separation (μm)	3.037
Number of fringe $\hookrightarrow$	37
Intersection volume diameter at $1/e^2$ intensity (μm)	81
Intersection volume length at $1/e^2$ intensity (μm)	774
Half intersection angle (degree)	5.98
Transfer constant ($\text{MHz}/\text{ms}^{-1}$)	0.329



All dimensions in mm

Figure 2.1 Dimensions of the 5-hole Pressure Probe

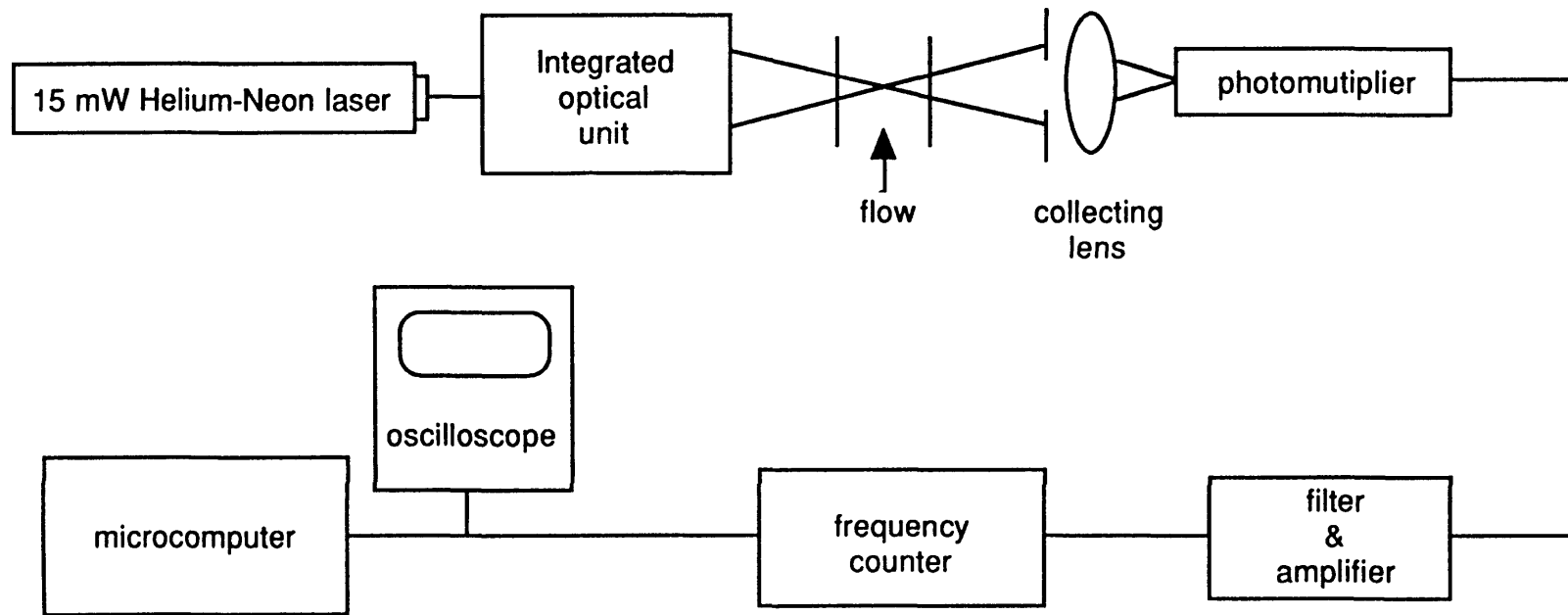


Figure 2.2 Diagram of the Laser-Doppler Velocimetry

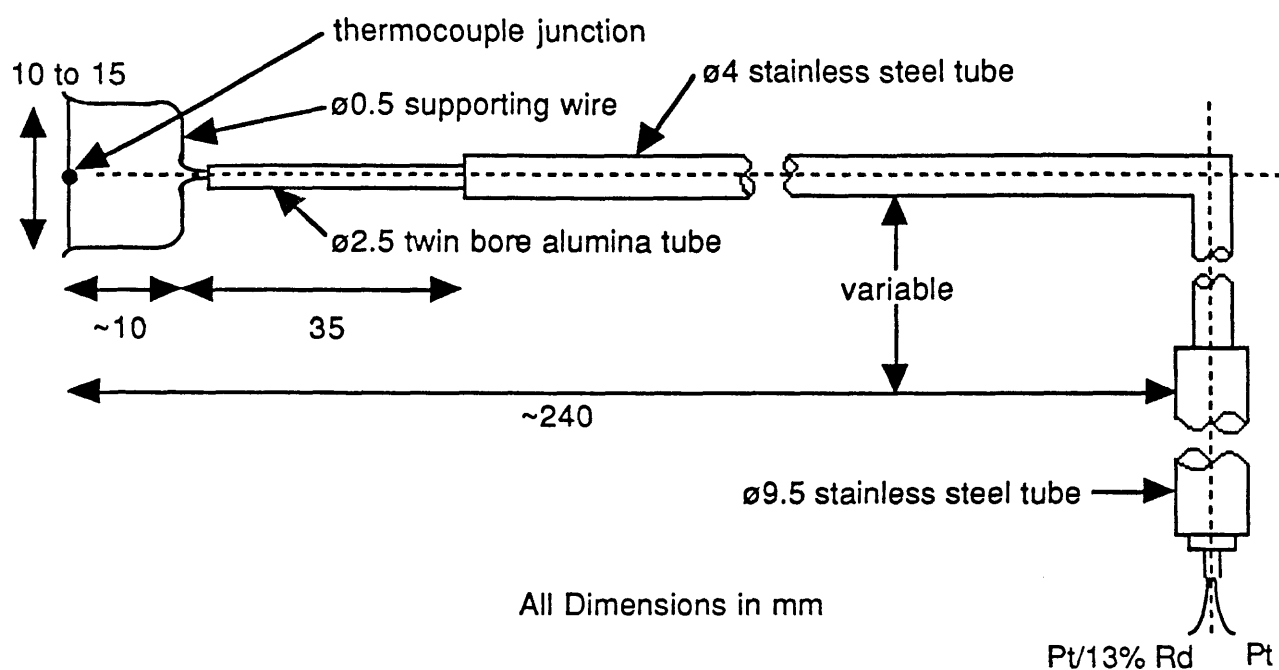


Figure 2.3 Dimensions of the Thermocouple Probe

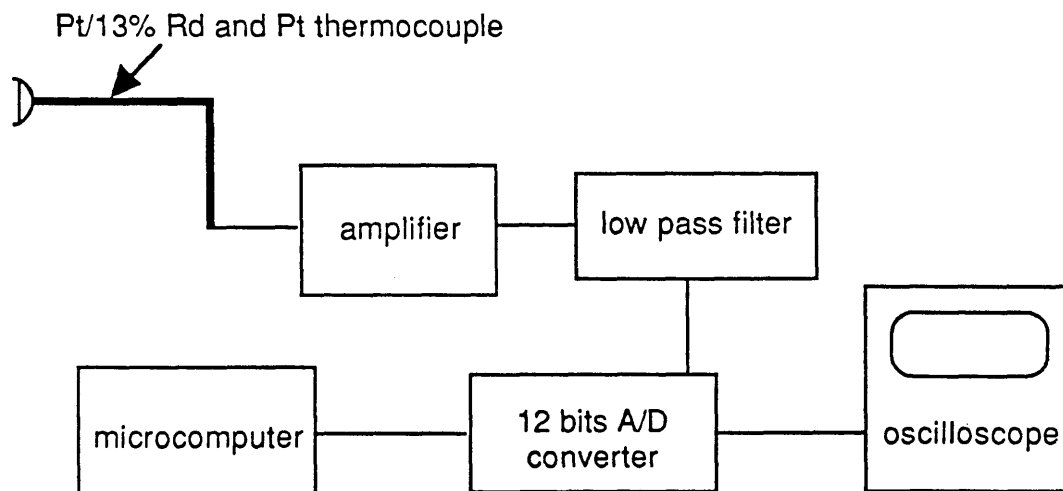


Figure 2.4 Schematic of the Thermocouple System

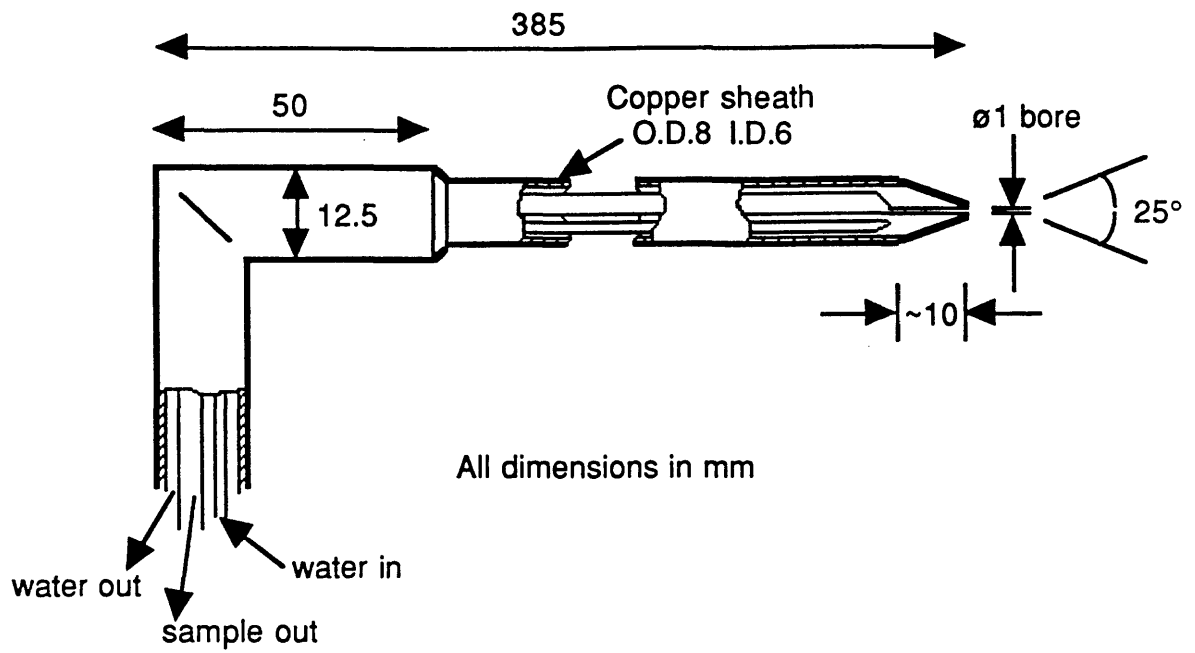


Figure 2.5 Dimensions of the Water-Cooled Probe

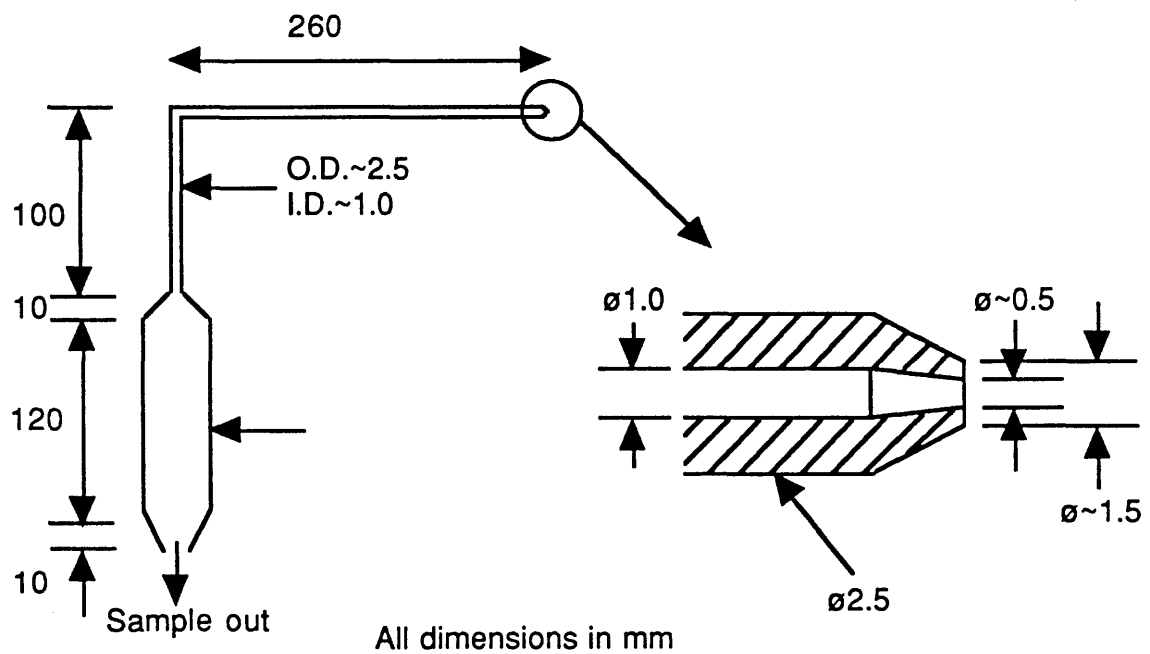


Figure 2.6 Dimensions of the Quartz Microprobe

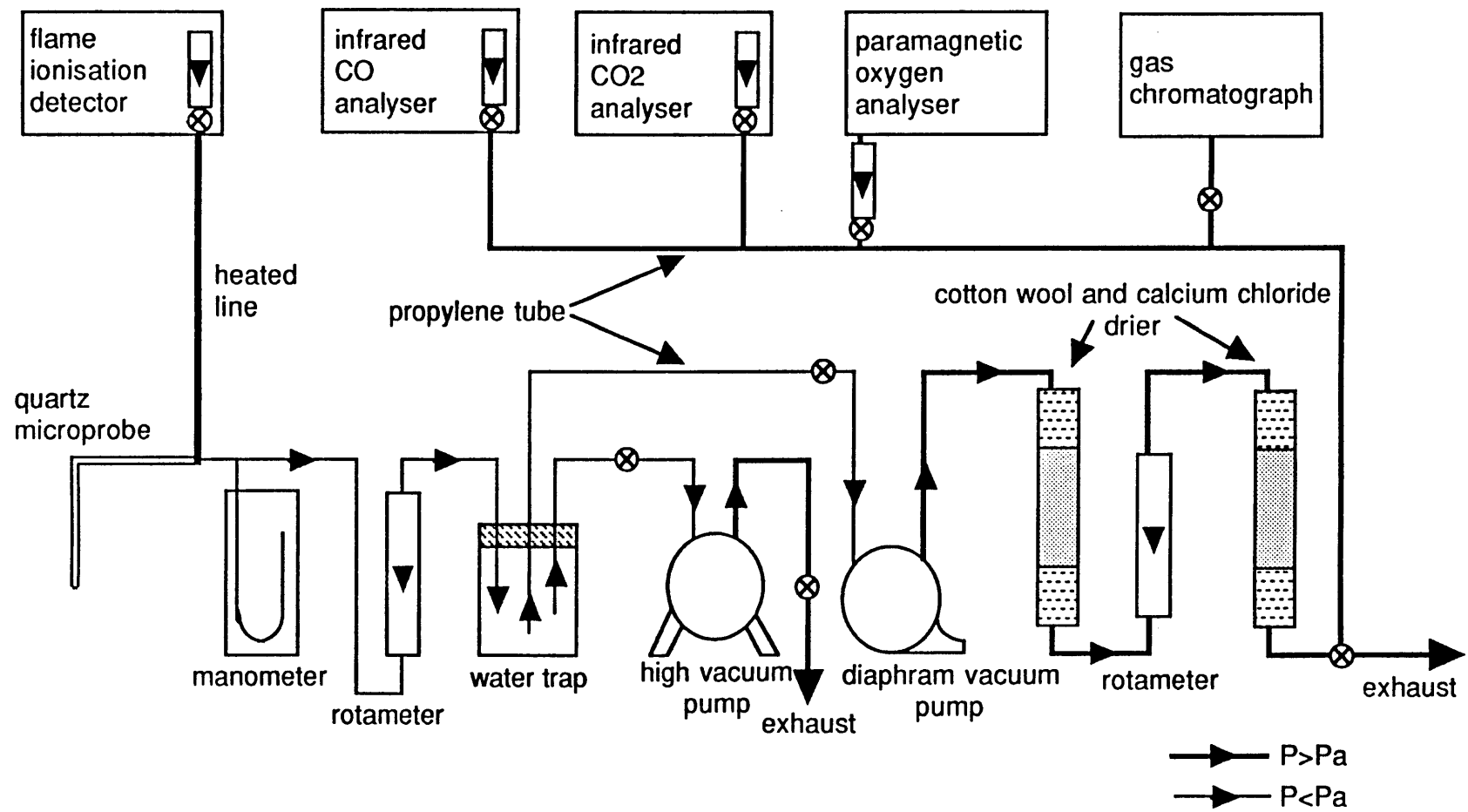


Figure 2.7 Schematic of the Sampling System of the Quartz Microprobe

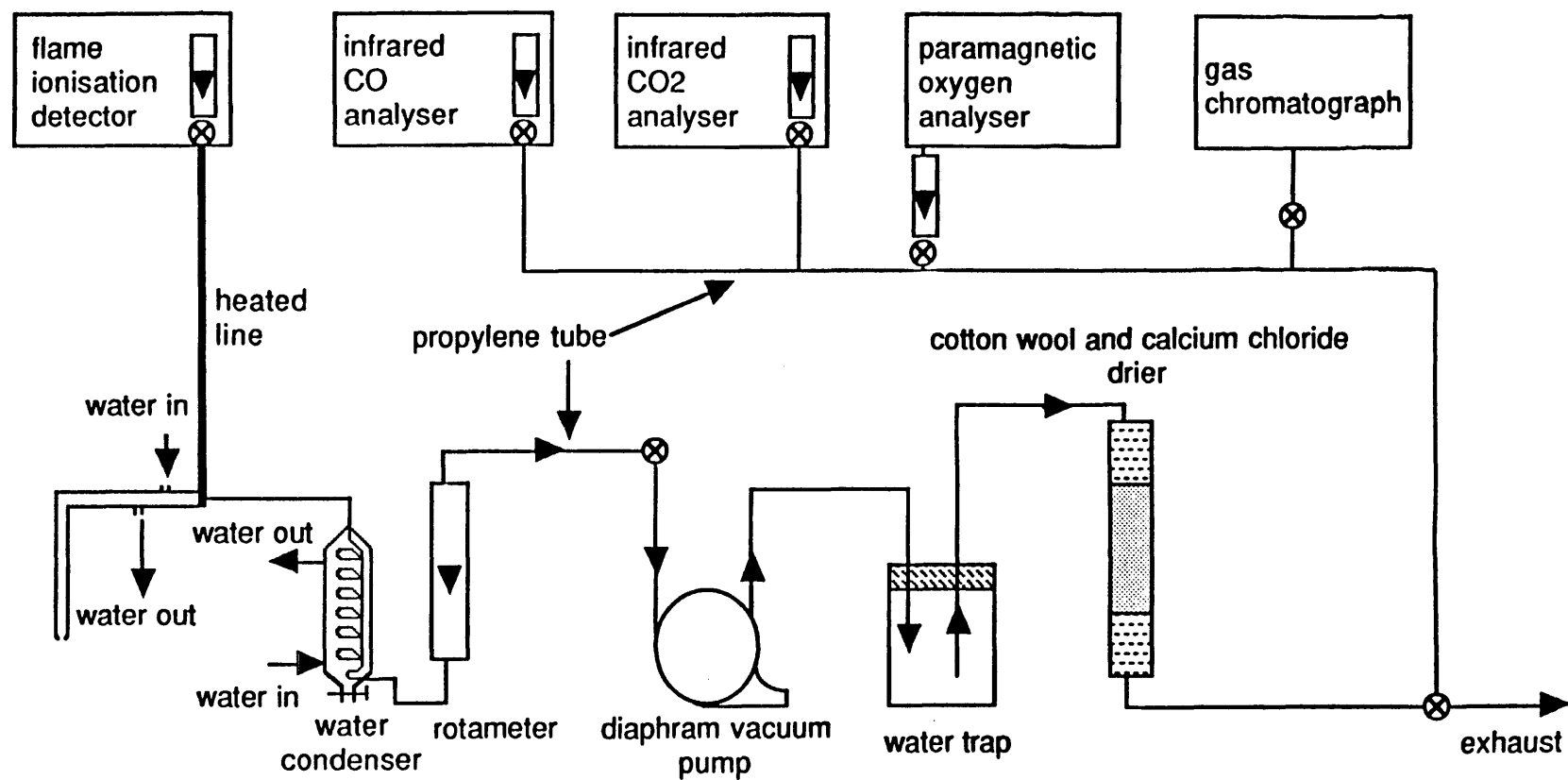


Figure 2.8 Schematic of the Sampling System of the Water-Cooled Probe

CHAPTER 3 CALCULATION PROCEDURE

3.1 Introduction

Numerical methods for the calculation of the properties of gas turbine combustor flows were reviewed in Chapter 1, where it was shown that their development has reached a stage that solutions can be obtained to supply results for preliminary analysis and the testing of combustor performance. The objective of this chapter is to describe the details of the calculation procedures used to predict the flowfields inside a curved duct and gas turbine combustor presented in Chapters 4 and 5 respectively.

The accuracy of various turbulence models and numerical techniques have been discussed in Section 1.2.3. The eddy viscosity turbulence model based on the solution of quantities for turbulent kinetic energy and dissipation rate represents a compromise in terms of accuracy and requirements of computational resources and is used in the calculations of Chapters 4 and 5. Higher order models may be superior for some flows, but cannot be justified in combusting flow calculations due to the large demands for computing time and memory. The modelling of combustion processes and turbulence/chemistry interaction in practical configurations remain the biggest problems in combustor prediction and no numerical procedure has yet been able to correctly represent the complex chemical characteristics inside the combustor, especially in the fuel rich regions in the primary zone. In the present investigation, equilibrium and laminar flamelet models have been tested with an assumed probability density function for a single scalar to represent the effect of turbulence on the combustion processes. Hybrid differencing, a combination of upwind and

central differencing, has been adopted for discretisation of the conservation equations.

The governing equations, turbulence and combustion models and the justification for their usage in the present study are given in the next section and Section 3.3 deals with the numerical discretisation procedures, the details of boundary conditions and methods of grid generation for the complex geometries concerned.

3.2 Description of the Equations

3.2.1 Exact Equations

The general equations governing the behaviour of Newtonian fluids can be expressed in cartesian tensor form as follows:

Continuity Equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho U_i) = 0 \quad (3.1)$$

Transport Equations for Momentum:

$$\frac{\partial}{\partial t}(\rho U_i) + \frac{\partial}{\partial x_j}(\rho U_i U_j) = \rho g_i - \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial U_i}{\partial x_j} - \rho u_i u_j \right) \quad (3.2)$$

Transport Equation for Scalar:

$$\frac{\partial}{\partial t}(\rho \Phi) + \frac{\partial}{\partial x_j}(\rho U_j \Phi) = \frac{\partial}{\partial x_j} \left(\Gamma \frac{\partial \Phi}{\partial x_j} \right) + \rho S(\Phi) \quad (3.3)$$

These equations represent a closed set but the range of time and length scales found in practical, high Reynolds number and turbulent flows is so wide that no existing computer can offer the required storage and speed for accurate solutions. The standard approach is to decompose the quantities characterising the flow into averaged values and fluctuating components with the time averaged quantities defined as:

$$\bar{\Phi} \equiv \lim_{N \rightarrow \infty} \frac{1}{N} \sum_{p=1}^N \Phi^{(p)} \quad (3.4)$$

$$\Phi = \bar{\Phi} + \Phi' \quad (3.5)$$

$$\overline{\Phi'} = 0 \quad (3.6)$$

In combusting and compressible flow, density-weighted averaging is favoured since the resulting equations are simpler and identical to the equations for time averaged properties in incompressible flow. The density-weighted mean values are given by:

$$\tilde{\Phi} \equiv \frac{1}{\bar{\rho}} \lim_{N \rightarrow \infty} \frac{1}{N} \sum_{p=1}^N \rho^{(p)} \Phi^{(p)} \quad (3.7)$$

and equations (3.1) to (3.3) may then be written in density-weighted averaged form as:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_i} (\bar{\rho} \tilde{U}_i) = 0 \quad (3.8)$$

$$\frac{\partial}{\partial t}(\overline{\rho U_i}) + \frac{\partial}{\partial x_i}(\overline{\rho U_i U_j}) = \overline{\rho} g_i - \frac{\partial \overline{P}}{\partial x_i} - \frac{\partial}{\partial x_j}(\mu \frac{\partial \widetilde{U_i}}{\partial x_j} - \overline{\rho u_i'' u_j''}) \quad (3.9)$$

$$\frac{\partial}{\partial t}(\overline{\rho \Phi}) + \frac{\partial}{\partial x_i}(\overline{\rho U_i \Phi}) = - \frac{\partial}{\partial x_i}(\Gamma \frac{\partial \widetilde{\Phi}}{\partial x_i} - \overline{\rho u_i'' \Phi''}) + \overline{\rho S(\Phi)} \quad (3.10)$$

Equations (3.8) to (3.10) are no longer a closed set with Reynolds stress and scalar flux correlations which require to be expressed in terms of known quantities. Although transport equations can be derived for these correlations, higher order correlations resulted as a closed set still cannot be obtained. The determination of the correlations is dealt with via a turbulence model which is discussed in the next section.

3.2.2 Turbulence Model

The second order correlations in equations (3.8) to (3.10), result from averaging the exact equations, must be modelled if a closed set is to be obtained. The modelling techniques can be classified into two broad groups: eddy viscosity and Reynolds stress models. Reynolds stress models are based on transport equations that govern the correlations so that nine extra equations, six for the stress correlations and three for the scalar flux correlations, have to be solved. However, the use of Reynolds stress models for calculation of combusting flows inside combustor geometries are premature in the context of limitations on computing storage and accuracy, as suggested by So, Whitelaw and Mongia (1988). Therefore, an eddy viscosity model, a relatively simple formulation to form a closed set and less demanding on computing resources, is favoured here.

The eddy viscosity models assume that the correlations are related to mean rate of strain through the introduction of the Boussinesq eddy viscosity concept and the correlations are defined as:

$$\overline{-\rho u_i u_j} = \mu_t \left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} (\bar{\rho} \tilde{k} + \mu_t \frac{\partial \bar{U}_k}{\partial x_k}) \quad (3.11)$$

$$\tilde{k} = \frac{1}{2} \overline{u_i u_i} \quad (3.12)$$

and

$$\overline{-\rho u_i \Phi} = \Gamma_t \frac{\partial \bar{\Phi}}{\partial x_i} \quad (3.13)$$

$$\Gamma_t = \frac{\mu_t}{\sigma_t} \quad (3.14)$$

where μ_t , σ_t and δ_{ij} are the eddy viscosity, turbulent Prandtl number and the δ_{ij} Kronecker delta respectively.

There are a number of models for describing the eddy viscosity, but the most popular and versatile is the two-equation k - ϵ model, introduced by Jones (1971) and Jones and Launder (1972), which is cast in density-weighted form as,

$$\mu_t = C_\mu \bar{\rho} \frac{\tilde{k}^2}{\tilde{\epsilon}} \quad (3.15)$$

$$\frac{\partial}{\partial x_j}(\rho \widetilde{U_j k}) = \frac{\partial}{\partial x_j} \left(\frac{\mu_t}{\sigma_k} \frac{\partial \widetilde{k}}{\partial x_j} \right) - \overline{\rho u_i u_j} \frac{\partial \widetilde{U_i}}{\partial x_j} - \frac{\mu_t}{\rho} \frac{\partial \overline{\rho}}{\partial x_j} \frac{\partial \overline{P}}{\partial x_j} - \overline{\rho \epsilon} \quad (3.16)$$

$$\frac{\partial}{\partial x_j}(\rho \widetilde{U_j \epsilon}) = \frac{\partial}{\partial x_j} \left(\frac{\mu_t}{\sigma_\epsilon} \frac{\partial \widetilde{\epsilon}}{\partial x_j} \right) - C_{\epsilon 1} \frac{\widetilde{\epsilon}}{\widetilde{k}} \left(\overline{\rho u_i u_j} \frac{\partial \widetilde{U_i}}{\partial x_j} + \frac{\mu_t}{\rho} \frac{\partial \overline{\rho}}{\partial x_j} \frac{\partial \overline{P}}{\partial x_j} \right) - C_{\epsilon 2} \overline{\rho} \frac{\widetilde{\epsilon}^2}{\widetilde{k}} \quad (3.17)$$

with ϵ as the rate of turbulent energy dissipation. The above equations contain some empirical constants which were obtained by comparing the model's performance with experimental data of two-dimensional shear flows, by Launder and Spalding (1974), and the values are:

$$C_\mu = 0.09 ; C_{\epsilon 1} = 1.44 ; C_{\epsilon 2} = 1.92 ; \sigma_k = 1.0 ; \sigma_\epsilon = 1.3$$

with a constant turbulent Prandtl number of 0.7. The above set of constants has also been tested by Rodi (1984) in other simple flows and shown to be valid with high turbulence Reynolds number which is found in most practical configuration and this version of the model constants was used in all the calculations reported here.

3.2.3 Combustion Model

In combusting flow calculations, the modelling of the combustion processes requires an assumption to deal with the source or sink term for the mass fraction of major species which appears in equation (3.10). Otherwise, the complicated elementary reaction kinetics and species formation rates, which are highly non-linear functions of species concentrations and temperature, have to be considered. For combustion models used in gas turbine combustor predictions, a single conserved scalar

approach is usually adopted since the heat loss is rarely greater than 1% of the heat release by reaction as suggested by Jones and McGuirk (1979). Thus, with low Mach number and equality of all diffusive transport coefficients, all thermochemical state variables can be expressed in terms of a single conserved scalar such as the mixture fraction or equivalence ratio.

Recent development of models to describe the turbulence/chemistry interaction to include finite rate effects has been made through the introduction of additional scalars describing the instantaneous chemical states. This, in turn, required the construction of a multi-dimensional joint probability density function, such as that described by Aguirre-Saldivar (1986), or involved assumptions such as statistical independence of the scalars as proposed by Correa (1985). However, these alternatives are computationally expensive and their usage was not strongly supported by experimental results as suggested by McGuirk (1987). Hence, the single conserved scalar approach was adopted in the combusting flow calculations in Chapter 5.

In the present study, two combustion models were tested. The equilibrium combustion model adopts a fast reaction assumption where turbulent mixing is assumed to control the reaction progress rather than chemical kinetics so that a mixture of fuel and oxidant instantaneously reach chemical equilibrium which was calculated by minimising the free energy as in the method of Gordon and McBride (1971). A rich flammability limit and the suppression of solid carbon, as suggested by Godoy (1982), was included. The equilibrium composition, temperature and gas density were then expressed as functions of the mixture fraction.

An alternative description of the thermochemical state can be achieved with the laminar flamelet approach as reviewed by Peters (1986). The model used in the present calculation here employed the global

reaction scheme by Jones and Linstedt (1988a) which approximated hydrocarbon flame structures with two reaction zones chosen to represent the breakdown of the fuel to intermediate carbon monoxide and hydrogen and the oxidation of these products to form water and carbon dioxide. This flamelet model has shown good agreement with experiments of laminar flames in relatively simple configurations as described by Jones and Linstedt (1988b). Its ability to predict the combustion characteristics of turbulent diffusion flames inside a combustor is discussed in Chapter 5.

In both combustion models, all thermochemical states were presumed known as unique functions of f , mixture fraction, on an instantaneous basis, that is,

$$\Phi_i = \Phi_i(f) \quad (3.18)$$

where Φ_i can be gas density, temperature or mole fraction of species. These relationships are commonly very non-linear and because of the fluctuations of the mixture fraction, the local mean value of any state variable must be calculated from,

$$\tilde{\Phi}_i = \int_0^1 \Phi_i(f) \tilde{p}(f) df \quad (3.19)$$

where $\tilde{p}(f)$ is the density-weighted probability density function, PDF, for f . The shape of the PDF can be assumed to take any form, such as a clipped Gaussian, beta-function or double delta-function distribution as discussed by Jones and Whitelaw (1982). The calculations by Jones (1980) showed that the use of double delta-function resulted in unrealistic temperature profiles while the difference between the clipped Gaussian and

beta-function was very little. In the present study, the beta-function distribution was adopted and the PDF is specified as:

$$\tilde{p}(f) = \frac{f^{a-1}(f-1)^{b-1}}{\int_0^1 f^{a-1}(f-1)^{b-1} df} \quad (3.20)$$

where a and b are given by:

$$a = \tilde{f} \left(\frac{\tilde{f}(1-\tilde{f})}{\tilde{f}^2} - 1 \right) \quad (3.21)$$

$$b = \frac{(1-\tilde{f})}{\tilde{f}} a \quad (3.22)$$

The variance of the mixture fraction which appears in equation (3.21) for the construction of the PDF is obtained from its own modelled transport equation:

$$\frac{\partial}{\partial x_j} (\rho \tilde{U}_j \tilde{f}^2) = \frac{\partial}{\partial x_j} \left(\frac{\mu_t}{\sigma_t} \frac{\partial \tilde{f}^2}{\partial x_j} \right) + 2 \frac{\mu_t}{\sigma_t} \left(\frac{\partial \tilde{f}}{\partial x_j} \right)^2 - C_D \rho \frac{\tilde{\epsilon}}{k} \tilde{f}^2 \quad (3.23)$$

where the last term is the modelled term for the mean scalar dissipation rate and introduces the model constant, C_D , which was assumed to have a value of 2.0, adopted by McGuirk (1987), in subsequent calculations. The mean density appears in the transport equation and may be obtained from this density-weighted PDF by:

$$\bar{\rho} = \left(\int_0^1 \frac{1}{\rho(f)} \tilde{p}(f) df \right)^{-1} \quad (3.24)$$

The above equations were used to describe the flow and combustion characteristics in a gas turbine combustor and the procedures taken to translate the equations into a numerically solvable form are discussed in the next section.

3.3 Methods for Calculations

3.3.1 Finite Difference Discretisation

The methodology adopted for solving the partial differential equations of Section 3.2 involves a standard finite difference method, as described by Patankar (1980). In general, all the transport equations can be represented in the general form:

$$\rho \frac{\partial \Phi}{\partial t} + \frac{\partial}{\partial x_i} (\rho U_i \Phi) = \frac{\partial}{\partial x_i} (\Gamma_\Phi \frac{\partial \Phi}{\partial x_i}) + \rho S_\Phi \quad (3.25)$$

where Φ can be averaged velocity or other scalar quantity. The two terms on the left hand side are the transient and convective terms while the gradient diffusion term and source term are on the right hand side. Γ_Φ is the appropriate diffusion coefficient. In all the calculations performed in the present study, no attempt was made to study the unsteady or transient effects of flow and combustion and, therefore, the transient term is omitted in subsequent discussion. The finite difference equation is derived by integrating equation (3.25) over a control volume and becomes:

$$\int_{\text{vol}} \left(\frac{\partial}{\partial x_i} (\rho U_i \Phi) \right) dV = \int_{\text{vol}} \left(\frac{\partial}{\partial x_i} \left(\Gamma_{\Phi} \frac{\partial \Phi}{\partial x_i} \right) \right) dV + \int_{\text{vol}} (\rho S_{\Phi}) dV \quad (3.26)$$

For simplicity, the discretised equation in one-dimension is:

$$\rho_i U_i \Phi_i - \rho_j U_j \Phi_j = \left(\Gamma_{\Phi} \frac{\partial \Phi}{\partial x} \Big|_i - \Gamma_{\Phi} \frac{\partial \Phi}{\partial x} \Big|_j \right) + \rho S(\Phi) \quad (3.27)$$

Two different methods for handling the dependent variables with respect to the position of the grid nodes can be used and are illustrated in Figure 3.1. The non-staggered grid method stores all variables at the grid nodes while the staggered grid arrangement stores the pressure and scalar quantities at grid nodes and the velocities between the scalar storage locations. With a pressure correction scheme, oscillatory pressure field can be produced with a non-staggered grid arrangement as suggested by Patankar (1980). This instability is caused by the second-order central differencing scheme which approximates the pressure gradient at each grid node with a spatial separation of $2\Delta x$ and hence the pressure difference between adjacent grid nodes cannot be predicted correctly. However, a staggered grid arrangement can overcome this instability but requires the adoption of different control volumes for the scalar and vector quantities. In the present study, both methods have been used and the oscillatory pressure field in non-staggered grid arrangement is overcome by the addition of a third order smoothing term in the continuity equation as by Rhie and Chow (1983).

In equation (3.27), the diffusion term is always approximated by central differencing:

$$\left(\frac{\partial \Phi}{\partial x}\right)_i = \frac{\Phi_W - \Phi_P}{\Delta x_W} \quad (3.28)$$

and the convective term by hybrid differencing,

$$\Phi_i = D_W \Phi_W + (1 - D_W) \Phi_P \quad (3.29)$$

which is a combination of upwind and central differencing and is chosen according to the cell Peclet number where:

$$D_W = 1/2 \quad \text{for } Pe \leq 2$$

$$D_W = 1 \quad \text{for } Pe > 2$$

and

$$Pe = \frac{U_i \Delta x_i}{\Gamma_\Phi} \quad (3.30)$$

The truncation errors associated with upwind and central differencing are of the order of Δx and $(\Delta x)^2$ respectively, where Δx is the grid spacing, as pointed out by Palma (1988). Central differencing, although a more accurate scheme, would lead to numerical instability when used in highly convective flows. The use of upwind differencing schemes can lead to a stable solution at the expense of a greater truncation error. The error is of the form of a 'false diffusion' which becomes more apparent when the direction of the flow is not in line with the local grid line.

The use of a higher order differencing scheme, like the QUICK formula of Leonard et al(1978) derived by fitting a parabola through three upstream weighted nodes, can further reduce the truncation error to the third or higher power of (Δx) . However, there is still no absolute evidence that higher order differencing gives a converged and physically realistic solution without numerical oscillation and there appears to be no absolute way of assessing numerical accuracy so as to determine which particular numerical scheme is superior, as suggested by So, Whitelaw and Mongia (1988). Hence, the hybrid differencing scheme was adopted in the calculations in Chapters 4 and 5.

The source term in equation (3.27) is then linearised as,

$$S(\phi) = S_U^\phi + S_P^\phi \phi_P \quad (3.31)$$

and the transport equation (3.25) for Φ re-arranged with the help of the continuity equation and generalised to three-dimensional form as:

$$\left(\sum_{i=1,N} a_i^\phi - S_P^\phi \right) \phi_P = \sum_{i=1,N} a_i^\phi \phi_i + S_U^\phi \quad (3.32)$$

where the summation terms extend over all neighbouring cell faces and N is equal to 6 in three-dimensional problems.

When the discretised transport equation is applied to a velocity U , the pressure gradient enters the equation as an unknown source term as:

$$\left(\sum_{i=1,N} a_i^U - S_P^U \right) U_P = \sum_{i=1,N} a_i^U U_i + A_W(P_W - P_P) \quad (3.33)$$

The solution of the above equation is required at every grid node for each dependent variable. The method adopted for solving the set of equations is an implicit, iterative technique, known as SIMPLE, proposed by Patankar and Spalding (1972). It first makes an initial guess for the pressure and velocity field and then proceeds with a guess and correct procedure for the pressure, using the most updated values for the variables to assemble the associated coefficients. This is done through a pressure correction equation which replaces the continuity equation and indicates the magnitude of the net mass flux inside a control volume. The iterative process continues until the pressure and velocity fields simultaneously satisfy all the transport equations and the pressure correction equation.

An under-relaxation factor is also introduced to the discretised equation which becomes:

$$\frac{1}{\alpha} \left(\sum_{i=1,N} a_i^\phi - S_P^\phi \right) \phi_P = \sum_{i=1,N} a_i^\phi \phi_i + S_U^\phi + \frac{1-\alpha}{\alpha} \left(\sum_{i=1,N} a_i^\phi - S_P^\phi \right) \phi_P^0 \quad (3.34)$$

where α is the under-relaxation factor and ϕ_P^0 is the value before iteration. This is necessary, because of the coupling and non-linearity of the equations, to ensure the convergence. The set of equations is then solved by a tri-diagonal matrix algorithm as described by Roache (1976).

The iteration continues until the criterion for convergence is satisfied. The parameter used to assess the convergence of the calculations is the residual term,

$$\phi_{res} = \frac{\sum_{i=1,N} a_i^\phi \phi_i + S^\phi - a_p^\phi \phi_p}{\phi_{max} - \phi_{min}} \Big|_{max} \quad (3.35)$$

which is the absolute value of the local continuity error where $\phi_{\max}$ and $\phi_{\min}$ are the maximum and minimum values of ϕ in the flowfield. The use of the difference between variables at consecutive iterations as a convergence criterion is not employed since it may indicate only that the solution changes slowly.

3.3.2 Boundary Conditions

The transport equations discussed in previous sections are elliptic and require values of each dependent variable or its gradient on all the boundaries of the solution domain. The following paragraphs describe the specifications for the different types of boundary encountered in the calculations.

(a) Inlet Boundary

The values for each dependent variable are specified for every grid node at the inlet plane. However, in the complex geometry of the combustor, the details of the mean and fluctuating properties are often unknown and the boundary conditions sometimes have to be estimated.

(b) Outlet Boundary

For an outlet boundary, zero gradient boundary condition is assumed in the direction normal to the outlet plane, i.e.

$$\Phi_{n-1} = \Phi_n \quad (3.36)$$

for any dependent variable, Φ , where there are n nodes in the direction normal to the outlet plane and the n th node represents the outlet plane.

(c) Plane of Symmetry

At a plane of symmetry, both the diffusion and convection fluxes, for all dependent variables, normal to the boundary are set to zero.

(d) Cyclic Boundary

The cyclic boundary is applied through the coupling of values of dependent variables, in any one co-ordinate direction for which cyclic boundary condition is applied,

$$\begin{aligned}\Phi_1 &= \Phi_m \\ \Phi_2 &= \Phi_{m+1}\end{aligned}\tag{3.37}$$

where there are $m+1$ grid points along that direction.

(e) Wall Boundary

The turbulence model described above is appropriate to high Reynolds number flows. For regions close to a solid wall, the small scale turbulence is no longer isotropic and the local Reynolds number of turbulence is so small that viscous effects dominate turbulent ones. It is possible to resolve the flow characteristics for the near wall region by using closely spaced grid nodes to resolve the gradients and by modifying the turbulence model such as that of Jones and Launder (1973). However, this would incur a high penalty in computational cost and, therefore, the alternative method for handling wall boundary, the wall function as described by Launder and Spalding (1974), has been used in the

calculations reported in this thesis. The empirical law of the wall, as in Schlichting (1968), is given by,

For $0 \leq y^+ \leq 11.5$, viscous sub-layer,

$$U^+ = y^+ \quad (3.38)$$

For $11.5 \leq y^+$, inertial sublayer,

$$U^+ = \frac{1}{\kappa} \ln (E y^+) \quad (3.39)$$

where E is an empirical constant, κ is the von Karman constant and

$$U^+ = \frac{U}{\left(\frac{\tau_w}{\rho}\right)^{1/2}} \quad (3.40)$$

$$y^+ = \frac{\rho y \left(\frac{\tau_w}{\rho}\right)^{1/2}}{\mu} \quad (3.41)$$

for which U is the velocity parallel to the wall, y is the perpendicular distance from the wall and τ_w is the wall shear stress. This empirical law is implemented through a modified version by Launder and Spalding (1974) and is given as,

$$\frac{\rho U C_\mu^{1/4} k^{1/2}}{\tau_w} = \frac{1}{\kappa} \ln (E y^+) \quad (3.42)$$

where

$$y^+ = \frac{\rho y C_\mu^{1/4} k^{1/2}}{\mu} \quad (3.43)$$

which would reduce to the above relation under local equilibrium condition.

The generation of turbulence kinetic energy is modelled through the source term of its transport equation for which,

$$S_u^k = \frac{\rho U C_\mu^{1/4} k}{y \ln(E y^+)} \quad (3.44)$$

The values of energy dissipation rate are estimated using the definition of turbulence length scale near the wall and the dissipation of a local equilibrium region, as mentioned by Gibson et al (1984), where

$$l = \frac{k^{3/2}}{\varepsilon} = -C_\mu^{-3/4} \kappa y \quad (3.45)$$

so that,

$$\varepsilon_w = \frac{C_\mu^{3/4} k^{3/2}}{\kappa y} \quad (3.46)$$

Furthermore, all the velocities and the gradient of energy dissipation rate are set to zero at a surface.

3.3.3 Grid Generation

For practical configurations, special numerical procedures are required to represent the complex geometries of the flow systems. Generally, there are two methods to deal with irregularly shaped boundaries and they are discussed below.

(a) Castellated Approximation

This method generates a regular orthogonal mesh which is overlayed to the flow domain and the discretised partial differential equations retain the form discussed above. The solid boundaries within the regular mesh are implemented by inactivating the cells which lie outside the flow domain through modifications in the finite difference equations. The modification can also set the values of dependent variable at selected grid nodes by changing the source terms in equation (3.32) to,

$$\begin{aligned} S_u^\phi &= G\Phi_G \\ S_p^\phi &= -G \end{aligned} \tag{3.47}$$

where G is a large number, say 10^{30} , and Φ_G is the appropriate values of dependent variables at the boundaries. Then equation (3.32) becomes,

$$\left(\sum_{i=1,N} a_i^\phi + G \right) \Phi_P = \sum_{i=1,N} a_i^\phi \Phi_i + G\Phi_G \tag{3.48}$$

and hence Φ_P is equal to Φ_G in the limit of G tends to infinity.

This method has previously been used in two-dimensional geometries by Koutmos (1985) and Palma (1988) and is extended to account for full

three-dimensional geometries in the present work. The results presented in Chapter 5 made use of this procedure to represent the boundaries of the combustor and the T-vaporiser which is fully three-dimensional. The disadvantages in this method are lack of flexibility and wastage of computing resources since nodes outside the flow domain are ignored.

(b) Body-Fitted Co-ordinates System

The limitations of the castellated approximation can be overcome by the use of a general curvilinear body-fitted co-ordinate system where the boundaries of the computational mesh are coincident with the physical boundaries. This method, in two-dimensional form, starts with the two Poisson equations in the physical plane, as described by Thompson (1980),

$$\begin{aligned}\frac{\partial^2 \eta_1}{\partial x^2} + \frac{\partial^2 \eta_1}{\partial y^2} &= P \\ \frac{\partial^2 \eta_2}{\partial x^2} + \frac{\partial^2 \eta_2}{\partial y^2} &= Q\end{aligned}\tag{3.49}$$

where η_1 and η_2 are the co-ordinates in the transformed plane and P and Q are control functions which may be used to alter the spacing of grid line inside the domain. The corresponding equation in the transformed plane can be obtained through the transformations:

$$\begin{aligned}\frac{\partial \eta_1}{\partial x} &= \frac{1}{J} \frac{\partial y}{\partial \eta_2} & \frac{\partial \eta_2}{\partial x} &= -\frac{1}{J} \frac{\partial y}{\partial \eta_1} \\ \frac{\partial \eta_1}{\partial y} &= -\frac{1}{J} \frac{\partial x}{\partial \eta_2} & \frac{\partial \eta_2}{\partial y} &= \frac{1}{J} \frac{\partial x}{\partial \eta_1}\end{aligned}\tag{3.50}$$

where

$$J = \frac{\partial x}{\partial \eta_1} \frac{\partial y}{\partial \eta_2} - \frac{\partial x}{\partial \eta_2} \frac{\partial y}{\partial \eta_1} \quad (3.51)$$

Then, in the transformed plane,

$$\gamma \frac{\partial^2 x}{\partial \eta_1^2} + \alpha \frac{\partial^2 x}{\partial \eta_2^2} - 2\beta \frac{\partial^2 x}{\partial \eta_1 \partial \eta_2} + J^2 \left(P \frac{\partial x}{\partial \eta_1} + Q \frac{\partial x}{\partial \eta_2} \right) = 0 \quad (3.52)$$

$$\gamma \frac{\partial^2 y}{\partial \eta_1^2} + \alpha \frac{\partial^2 y}{\partial \eta_2^2} - 2\beta \frac{\partial^2 y}{\partial \eta_1 \partial \eta_2} + J^2 \left(P \frac{\partial y}{\partial \eta_1} + Q \frac{\partial y}{\partial \eta_2} \right) = 0$$

where

$$\alpha = \left(\frac{\partial x}{\partial \eta_1} \right)^2 + \left(\frac{\partial y}{\partial \eta_1} \right)^2 \quad (3.53)$$

$$\beta = \frac{\partial x}{\partial \eta_1} \frac{\partial x}{\partial \eta_2} + \frac{\partial y}{\partial \eta_1} \frac{\partial y}{\partial \eta_2} \quad (3.54)$$

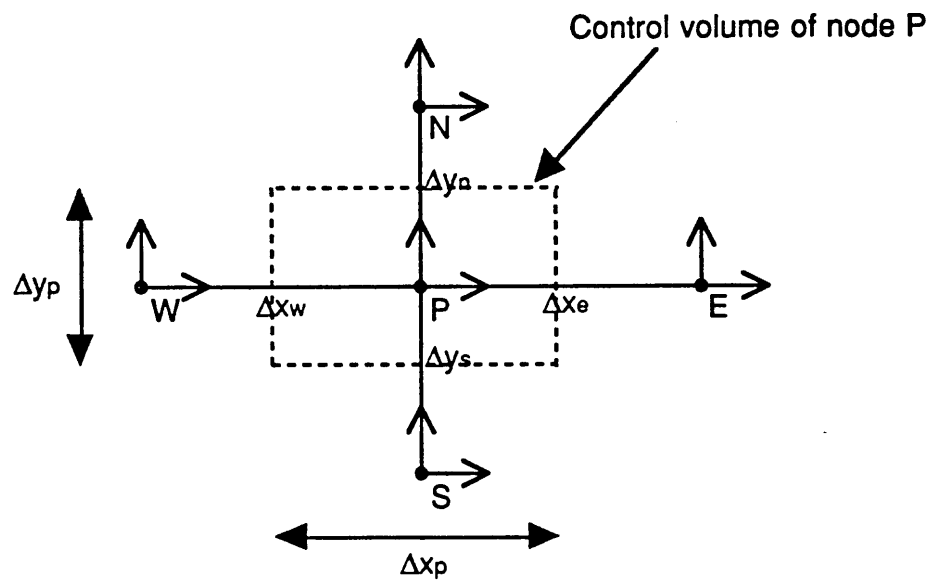
$$\gamma = \left(\frac{\partial x}{\partial \eta_2} \right)^2 + \left(\frac{\partial y}{\partial \eta_2} \right)^2 \quad (3.55)$$

In the present study, P and Q are set to zero and $\Delta \eta_1 = \Delta \eta_2 = 1$. Equation (3.50) are then discretised by standard central differencing techniques. The equations are then solved iteratively by a line-Gauss-Seidel method incorporating the Thomas algorithm.

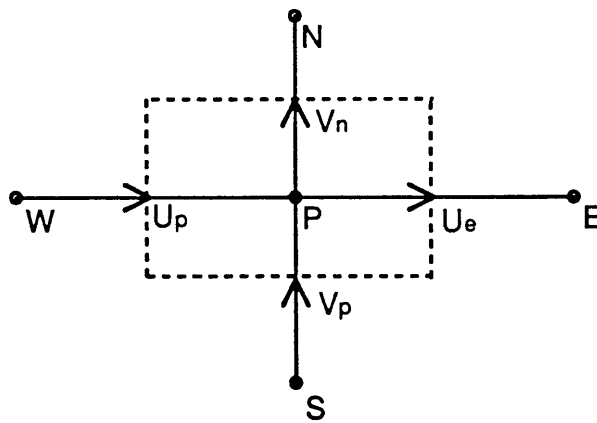
The mesh generated would correspond to a curvilinear co-ordinate system and the governing equations are transformed to this system according to,

$$\begin{aligned}\frac{\partial \Phi}{\partial x} &= \frac{1}{J} \left(\frac{\partial y}{\partial \eta_2} \frac{\partial \Phi}{\partial \eta_1} - \frac{\partial y}{\partial \eta_1} \frac{\partial \Phi}{\partial \eta_2} \right) \\ \frac{\partial \Phi}{\partial y} &= \frac{1}{J} \left(\frac{\partial x}{\partial \eta_1} \frac{\partial \Phi}{\partial \eta_2} - \frac{\partial x}{\partial \eta_2} \frac{\partial \Phi}{\partial \eta_1} \right)\end{aligned}\tag{3.56}$$

Boundary conditions are applied in a similar manner to that of the regular mesh method and this body-fitted mesh technique has been used in the calculations of flow in the curved duct in Chapter 4.



(a) Non-Staggered Grid Arrangement



(b) Staggered Grid Arrangement

Figure 3.1 Methods of Grid Arrangement

CHAPTER 4 FLOW IN A CURVED DUCT

4.1 Introduction

This chapter describes the experimental and numerical results obtained in a curved U-shaped duct in order to provide more detailed understanding of the flow through the curved exit nozzle, found in the Gem combustor, which directs the flow from the dilution zone of the combustor to the inlet guide vane of the turbine. The perspex model provided a more uniform entry profile to the nozzle and, therefore, the effect of the curvature can be isolated from that of the variation of inlet velocity profiles with better optical access to the flow domain. The investigation was necessary because previous studies of curved duct or pipe, as mentioned in Chapter 1, have been limited to relatively mild curvature and there has been no measurement in a curved nozzle with radius-to-width and inlet-to-exit area ratios of relevance to those of the nozzle found in the reverse-flow combustor. The calculations performed for the same geometry would quantify the extent to which flow characteristics can be represented with existing methodology.

The measurement techniques and the associated errors have been discussed in Chapter 2. Two orthogonal components of mean velocity and the corresponding normal stresses were measured with a laser-Doppler velocimeter in forward scattering mode with seeding, in the form of small silicone oil droplets, supplied through an atomiser and the Doppler signals processed by a frequency counter. Pressure was measured on the side wall and at the two curved surfaces in the plane of symmetry.

Numerical simulations were made with the procedures described in Chapter 3 and involved the standard finite difference method with the two-equation k - ϵ turbulence model. A general curvilinear co-ordinate system, as described in detail in Section 3.3.3 (b), based on a non-orthogonal boundary conforming transformation was adopted. Three computational meshes have been used to quantify the effect of grid dependence.

The general nature of flows through curved ducts have been described in Chapter 1. For turbulent flows through ducts with sharp curvature, the streamwise velocity distributions are expected to be skewed towards the inner wall, at least in the first half of the duct, as observed by Brown and Morris (1963), Taylor, Whitelaw and Yianneskis (1981) and Humphrey, Whitelaw and Yee (1981). This is contrary to the situation where the flow is laminar or in a duct of mild curvature, as shown by Taylor, Whitelaw and Yianneskis (1981) and Hille et al (1983). This is due to the separate effects of pressure gradient, as a result of the curved geometries, and secondary flow and their respective influences will be examined with respect to the results presented in this chapter and previous measurements.

The next section introduces the flow configuration and the details of calculations are described in Section 4.3. The experimental and numerical results are presented in Section 4.4 and discussed in Section 4.5. Section 4.6 provides a summary of conclusions.

4.2 Flow Configuration

The flow arrangement, Figure 4.1, consisted of a centrifugal fan supplying air at a constant rate through a diffuser and a contraction to the test section. A straight rectangular duct, with cross section of 34mm x

130mm and 500mm in length, was used to connect the contraction and the curved test section to ensure a uniform and symmetric inlet profile. The geometry for the perspex, U-shaped, curved test section, Figure 4.2, retained the curvature of the exit nozzle of the combustor with inside and outside radius of 16mm and 40.5mm respectively and the centre of curvature of the two curved surfaces was offset by 9.5mm to give an inlet-to-exit area ratio of 2.3 and the radius-to-width ratio, based on the mean radius of curvature and width, is 1.2.

Measurements were made with bulk mean inlet velocities of 4.41m/s and 11.77m/s corresponding to Reynolds numbers of 15,000 and 40,000 respectively based on a hydraulic diameter of 54mm. The higher Reynolds number flow would then quantify the effect of the inlet Reynolds number to the flow development. Preliminary measurements were obtained at locations along the straight upstream section to confirm the symmetry of flow to the curved section.

4.3 Details of Calculations

Calculations were performed for the geometry of the exit nozzle with a bulk mean inlet velocity of 4.41m/s. The inlet boundary conditions were taken from velocity measurements at a location one hydraulic diameter upstream of the entry to the curved part of the nozzle where the measurements showed no upstream influence of the curvature of the bend to the local flowfield. The dissipation rate was estimated by the relation,

$$\epsilon = \frac{k^{3/2}}{L} \quad (4.1)$$

with L , the characteristic length scale, equals to the hydraulic diameter and k is the measured mean turbulence intensity at the location one hydraulic diameter upstream of the entry. The three computational meshes of Figure 4.3 were used with the coarse grid containing $32 \times 10 \times 25$ nodes in the streamwise, radial and transverse directions respectively, the standard grid $58 \times 15 \times 25$ nodes and the fine mesh $101 \times 20 \times 35$ nodes. The fine mesh required a computer memory of 7 Mbytes to run and was the finest mesh that could be used with the existing computing facilities available. The conservation equations of velocity, pressure correction, turbulence kinetic energy and dissipation rate were transformed to a curvilinear co-ordinate system and then solved in the manner described in Chapter 3. The boundary conditions for the inlet and outlet planes, the three solid walls and the plane of symmetry were applied in the manner described in Section 3.3.

4.4 Results

4.4.1 Measurements

Figure 4.4 shows the co-ordinate system adopted for the presentation of measurements in the curved duct with Z^* the non-dimensional distance from the plane of symmetry in the spanwise direction and R^* the distance in the radial direction, non-dimensionalised by the local duct width, from the inner wall. Preliminary measurements of mean streamwise velocity at locations in the straight part of the test section were made, Figure 4.5, to confirm the symmetry of the inlet velocity profiles to the curved duct. The velocities of the core fluid, $-0.8 < Z^* < 0.8$, have increased slightly, about 0.7% of the bulk mean velocity, from a location 2.5 hydraulic diameter upstream of the entry, $X_H = -2.5$, to $X_H = -1.5$. This is because the flow was

not fully-developed at these locations and the boundary layer continued to grow, as observed in Figure 4.5, and led to lower mass flow in the near wall regions and higher mass flow in the core region. The mean velocity profiles also indicate that the flow upstream of the inlet was symmetric within 3% of the mean velocities with no discernable trend. Therefore, results are presented in half cross-planes at downstream planes.

Measurements of mean and rms of two orthogonal components of velocity were obtained at six planes along the test section corresponding to: one hydraulic diameter upstream of entry to the curved part of the duct ($X_H = -1.0$), 0° (entry), 45° , 90° , 135° and 180° (exit) with distance between locations of measurements of 5mm in the z-direction (2.5mm in the near wall regions for $Z^* > 0.7$) and 4mm in the radial direction (3mm at the 180° plane). At 20mm downstream of the exit, probe measurements were possible and a five-hole probe was used to record the three components of mean velocity. Wall static pressure measurements were measured through wall tapings at the locations of the LDV measurements and registered by a micro-manometer in the wall of the test section.

Figure 4.6 shows the mean and rms streamwise velocity contours, non-dimensionalised by the local bulk mean velocity, at the upstream station of the curved nozzle. Negligible cross-stream velocities were recorded. The distribution of velocity is uniform and symmetric, with respect to $R^* = 0.5$. The displacement boundary layer thickness is 4mm and the rms velocity fluctuations are smaller than 5% of the bulk mean velocity in the core and increase to 10% near the boundary. The extent of the boundary layer region indicates that the flow is not fully-developed at this location which is about 10 hydraulic diameter downstream of the exit of the contraction. The contours also indicate that the influence of the bend is absent in this plane.

The mean and rms velocity fluctuations for the streamwise and radial components, non-dimensionalised by the local bulk mean streamwise velocity, $U_{b,\theta}$ (θ denotes the angle between the plane of measurement and the inlet plane of the duct), are shown in Figure 4.7 for the 0° plane (entry) of the curved section. The core fluid has been displaced towards the inner wall with the maximum mean streamwise velocity of $1.3 U_{b,0}$ at $R^*=0.15$. This represents a stronger radial pressure gradient towards the inner wall, as compared to the study of Chang et al in a 180° duct (maximum streamwise at the 3° plane was about $1.2 U_b$ at $R^*=0.3$) due to the stronger curvature. Velocities decrease gradually towards the outer wall with magnitude of $0.7 U_{b,0}$ at $R^*=0.8$ and the distributions of radial velocity show that fluid is directed towards the suction surface, inner wall, with a maximum of $0.25 U_{b,0}$ in the central zone and at the side wall with values of lower than $0.1 U_{b,0}$ for $R^*>0.8$. This high cross-stream velocity confirms the strong radial pressure gradient caused by the sharp curvature of the bend. A region of weaker radial velocity, with magnitude of $0.17 U_{b,0}$, is observed between $Z^* = 0.7$ and 0.8 and also close to the side wall for $Z^*>0.9$. This represents the extent of influence of the secondary flow where fluid is transported from the pressure surface to the suction surface close to the side wall and the region is limited to only about one third of the width of the channel due to its large aspect ratio in the present study. Streamwise velocity fluctuations up to $0.15 U_{b,0}$ occur close to the walls with smaller values, over $0.08 U_{b,0}$ for the radial component.

At the 45° plane, the core of the fluid is displaced further towards the inner wall with a maximum streamwise velocity greater than $1.3 U_{b,45}$ for $R^*<0.2$ whereas the magnitude near the outer wall is reduced to below $0.7 U_{b,45}$ for $R^*>0.7$, Figure 4.8. The radial velocities, with maximum of $0.2 U_{b,45}$ around $Z^*=0.8$ near the inner wall ($R^*<0.1$), show a change of

flow direction compared to that at the entry plane and the influence of the secondary flow in the near wall region can still be observed. The change of flow direction for the radial velocities is due, partly, to the short distance at the inner wall between the two measuring locations and the pressure gradient is insufficient to direct the high momentum fluid near the inner wall along the curvature of the suction surface and hence resulted in a local flow direction away from the inner wall. High values of velocity fluctuations are also observed, with maximum values of the order of $0.15 U_{b,45}$, near the pressure surface both in the streamwise and radial direction. Compared with the 0° plane, the distribution of fluctuations of streamwise velocity remains unchanged whereas, for the radial component, the levels of turbulence fluctuation increase near the pressure surface.

The development of velocity distributions further downstream is shown in Figures 4.9, 4.10 and 4.11 for the 90° , 135° and 180° plane respectively. The maximum streamwise velocity is reduced to $1.2 U_{b,90}$ close to the inner wall in the 90° plane compared with a value of above $1.3 U_{b,45}$ in the 45° plane and the distribution remains the same near the outer wall regions with magnitude of below $0.7 U_{b,90}$ for $R^* > 0.8$. The radial velocities towards the inner wall are about $0.1 U_{b,90}$ due to the greater radial pressure gradient as shown in Figure 4.14. For the 135° plane, the streamwise velocity retains its maximum value of $1.2 U_{b,135}$ at the inner wall whereas near the outer wall region, the magnitude is less than $0.9 U_{b,135}$ for $R^* > 0.8$. The acceleration of the fluid between the 90° and 135° plane near the outer wall region is due to the area contraction of the duct and the reduced radial pressure gradient between the two plane, Figure 4.14. The influence of the secondary flow near the side wall region can still be observed. The velocity fluctuations at the 90° and 135° planes are anisotropic with high radial velocity fluctuations near the pressure surface

while high streamwise velocity fluctuations are recorded near the suction surface as a result of the secondary motion in the flow domain, as suggested by Taylor, Whitelaw and Yianneskis (1981). The streamwise velocity at the 180° plane shows a more uniform distribution with maximum of $1.06 U_{b,180}$ near the inner wall and values of about $0.96 U_{b,180}$ at $R^*=0.8$. The radial velocity distribution shows a fluid motion towards the outer wall due to the diminishing radial pressure gradient near the exit of the curved duct.

Figure 4.12 shows the distributions of the three components of mean velocity at the exit of the curved duct (20mm downstream of the 180° plane) and Figure 4.13 a vector plot illustrating the effect of the secondary flow near the side wall. The streamwise velocity has a uniform distribution with values of about $0.8 U_{b,exit}$ at $Z^*=0.7$ and the velocity in the z-direction has its maximum of $0.1 U_{b,exit}$ close to the inner and outer wall for $Z^*>0.7$. The inward flow radial velocity has a maximum of $0.13 U_{b,exit}$ at the inner wall of the exit plane because the fluid coming out of the exit of the nozzle encountered a sudden expansion to the open air surrounding, the flow then reattached to the bottom wall of the straight section of the duct, Figure 4.2, and resulted in a small recirculation close to the inner wall. The low pressure within the recirculation, therefore caused the fluid flowed towards the inner wall.

The wall static pressure measurements are presented in Figures 4.14 and 4.15 at the plane of symmetry and the side wall respectively with the non-dimensionalised pressure coefficient, C_p , defined as,

$$C_p = \frac{P - P_{ref}}{\frac{1}{2} \rho U_b^2} \quad (4.2)$$

with P_{ref} the wall static pressure at $X_H = -1.0$ and $Z^* = 0$ and U_b the bulk mean inlet streamwise velocity. Initially, an adverse pressure gradient develops at the pressure surface up to the 45° plane, as in the case of a 90° bend with sharp curvature studied by Taylor, Whitelaw and Yianneskis (1981), and thereafter, the pressure decreases. A favourable pressure gradient can be seen at the suction surface causing the acceleration of the fluid towards the inner wall. The largest pressure difference is found at the 90° plane. The radial pressure gradient at the side wall retains a similar magnitude from the 45° to the 180° plane whereas at the entry, a smaller radial pressure gradient is registered.

A second flow condition, with inlet Reynolds number of 40,000, was employed to investigate the effect of inlet Reynolds number on the flow development inside the curved duct and results of velocity measurements are shown from Figures 4.16 to 4.21 in terms of isotachs for the mean and rms streamwise and radial velocity at six locations: one hydraulic diameter upstream of the entry to the curved duct, the 0° , 45° , 90° , 135° and 180° plane. The flow pattern remains essentially unchanged as compared to the lower Reynolds number flow but quantitative differences are observed and will be discussed in the discussion in Section 4.5.

4.4.2 Calculations

The calculations were performed on a VAX 8600 with a VAX/VMS operating system for the coarse and standard grid and the fine grid calculation was performed on an Amdahl 5890 mainframe computer due to its larger storage. The convergence criterion was the same as that described in Section 3.3.1 and the iteration process was terminated when the maximum residual of each dependent variable, normalised by the root-mean-square value of each variable, was less than 10^{-4} . The CPU times

needed for convergence were 45 minutes for the coarse grid and 120 minutes for the standard grid in the VAX 8600. For the fine grid calculation, about 1500 iterations was necessary with each iteration requiring about 40 seconds of CPU time on the Amdahl 5890. Results to show the effect of grid dependence are presented first and then the fine grid calculations are compared with the measurements presented in Section 4.4.1.

Figure 4.22 shows the predicted streamwise velocity profiles at three planes, 0° , 90° and 180° , at $Z^*=0.015$ and 0.985 (1mm from the side wall) for the three grids used. At $Z^*=0.015$, the velocity profiles with the three grids are within 2% of each other, except at the 180° plane where a difference of 6% is observed in the near wall region between the coarse grid and the other two. At $Z^*=0.985$, the results given by the coarse and standard grids are very close indicating that the increase in number of grid nodes by 66% and 50% in the streamwise and radial directions respectively has little effect on the development of flow in the near wall region. The fine grid results, with three and two times the number of nodes of the coarse grid in the streamwise and radial directions respectively, shows lower momentum flow rates in the near wall region and it can also be concluded that a grid independent solution cannot be obtained with the spatial resolution given by this grid. This is further confirmed by Figure 4.23 which shows the radial velocity profiles at the same location as in Figure 4.22. The maximum difference at $Z^*=0.015$ is about 10% whereas, in the near wall region, the discrepancies are more obvious and increasing as the flow developed along the curved duct.

The difference in the results predicted with the three grids is due mainly to numerical diffusion as a result of different grid densities since the physical models and numerical schemes adopted were the same for all

the calculations. The typical cell Peclet number, as defined in Section 3.3.1, was of the order of 10 for the fine grid in the streamwise component of velocity near the plane of symmetry and this implied the use of upwind differencing with first order accuracy. Since the flow was predominantly mesh-aligned, numerical diffusion errors were not expected to be severe and the results were grid independent as observed in Figures 4.22a to 4.22c and 4.23a to 4.23c. In addition, the flow structure was relatively simple and the difference with different grid resolution was not apparent. However, for the near wall region where the secondary flow effect was obvious, as discussed in Section 4.4.1, with a typical secondary velocity scale an order of magnitude smaller than the streamwise component and with the flow direction not aligned with the mesh, the discrepancies caused by numerical diffusion were likely to be more severe for the coarser grids and the results were, therefore, grid dependent. For the fine grid, the typical cell Peclet number was of the order of 1, which implied central differencing with second order accuracy. Hence, the predicted trends of flow development were different between the fine grid and the other two, in the near wall regions, as observed in Figures 4.22d to 4.22f and 4.23d to 4.23f. It can also be concluded that the computational meshes employed in the present study, at least for the coarse and standard grid, were unable to resolve the fine structure of the secondary flow in the near wall regions.

Figure 4.24 allows comparison between experiment and calculation for the streamwise velocity profiles at the 0° plane. The velocity distribution of the core flow can be predicted to within 7% with the maximum discrepancy found in the near wall regions. For $Z^* > 0.85$, however, the velocity profiles are underpredicted by up to 25% in the near wall regions. The corresponding predicted radial velocity profiles of Figure 4.25 do not show the influence of the secondary flow found in the

near wall region and the radial velocity distributions are almost identical across the width of the duct with maximum velocity of 0.7m/s toward the centre of curvature, whereas the measurements shows a minimum inward flow velocity at $Z^*=0.85$.

The streamwise and radial velocity distributions at the 45° plane are shown in Figures 4.26 and 4.27 respectively. Again, the streamwise velocity in the central regions of the duct can be represented by prediction to within 10% but the calculation is unable to predict the swirl-type flow as a result of the secondary flow effect. The comparison of streamwise velocity profiles in the 90°, 135°, 180° planes and 20mm downstream of the exit, are shown in Figures 4.28 to 4.31. The discrepancies increase as the flow develops along the duct and, at the 180° plane, a shift of the maximum velocity from the inner surface to the outer surface is predicted in the central region of the duct.

From the numerical results presented above, it can be concluded that the bulk flow through the duct were predicted to within 10% of the measurements where the flow was influenced mainly by pressure gradient and a grid independent solution could be obtained. However, in near wall regions where the influence of secondary flow dominated, the calculations failed to represent the flow development and solutions given by successive grid refinements were different. This is, in part, due to the assumption of the k- ϵ model since the present flow were strongly anisotropic but further grid refinement is likely to give a better description without modification to the turbulence model since a grid independent solution has not been achieved in the present case.

4.5 Discussion

The flow development inside curved geometry is under the separate influences of pressure gradient and cross-stream convection by the secondary flow as suggested by Taylor, Whitelaw and Yianneskis (1981). The pressure gradient causes an inward flow towards the suction surface, inner wall, whereas the secondary flow convects low momentum fluid near the pressure surface and along the side wall to displace the core fluid close to the inner wall. The measurements of Taylor, Whitelaw and Yianneskis (1981) showed that thinner boundary layers led to smaller cross stream velocities and, also, that the secondary flow was dependent on the inlet velocity distribution rather than the Reynolds number as suggested by Enayet et al (1981).

Since the upstream tangent to the present curved bend was of fixed length, the effect of increasing the Reynolds number was to decrease the boundary layer thickness in the entry plane of the bend. The difference in the inlet profiles between the two cases can be observed when comparing Figures 4.6 and 4.16. Although the boundary layer thickness at the two curved surface are the same for both cases, the velocity in the near side-wall region increased with Reynolds number, e.g. at $Z^*=0.9$ and $R^*=0.5$, the velocity is $0.95 U_{b,inlet}$ as compared to $0.8 U_{b,inlet}$ for the lower Reynolds number case. This relatively fast moving fluid near the side wall for the higher Reynolds number flow reduced the effect of the secondary flow which transported the lower momentum fluid by cross stream velocity towards the suction surface along the side wall. This led to a higher maximum streamwise velocity for the higher Reynolds number flow as observed by comparing Figures 4.7 and 4.17. The magnitude of streamwise velocity near the suction surface was about 10% higher for the

case of higher Reynolds number flow whereas the distribution of the velocity fluctuation remained the same. The radial velocity distribution also remained the same for the two cases indicating the influence of the pressure gradient to the flow development was the same. This agrees with the finding of Azzola et al (1986) where, for flow through a 180° curved pipe, the dimensionless profiles was essentially independent of the Reynolds number for the same non-dimensionalised inlet velocity profiles.

At further downstream regions, the differences in the streamwise velocity profiles diminished as observed by comparing Figures 4.8 to 4.11 and 4.18 to 4.21. The similarity between the two sets of results suggests that the influence of boundary layer thickness, and hence the strength of the secondary velocity, diminished as the flow developed through the curved channel and this is supported by the results of Taylor, Whitelaw and Yianneskis (1981).

The effect of the sharp curvature of the duct in the present case, with radius-to-width ratio of 1.2, can be observed when the results are compared with those of Chang et al (1983) in a square-sectioned curved duct with radius-to-width ratio of 3.35 and with the earlier results of Brown and Morris (1963), using pressure probes, in a curved duct with ratio of 1.83. In all three cases, the streamwise velocity showed their maximum values displaced towards the inner wall between the entry and 45° plane. The secondary flow was more intense from the 45° plane in the case of relatively mild curvature studied by Chang et al (1983) and the position of maximum streamwise velocity shifted towards the outer wall and strong radial velocity, about $0.1 U_b$, towards the inner wall was found near the side wall. This phenomenon was also observed in a 90° bend of Humphrey, Whitelaw and Yee (1981) and Taylor, Whitelaw and Yianneskis (1981).

The flow development in the present study was more in agreement with the observation of Brown and Morris (1963) where the position of maximum streamwise velocity remained close to the inner wall throughout the curvature of the duct. For the radius ratio of 1.83, the maximum streamwise velocity attained a value of about $1.2 U_b$ at the 40° plane and thereafter, this maximum value gradually decreased along the duct whereas in the present case, the maximum value is $1.3 U_b$ at the 45° plane and retained this value up to the 135° plane. This is due to, first, the sharper curvature and therefore a stronger radial pressure gradient and, secondly, the contraction in cross-sectional area which led to greater streamwise momentum and hence the effect of the secondary flow was reduced.

The calculations performed here was based on the elliptic form of the conservation equations which removes the assumption of the absence of recirculation in the planes parallel to the streamwise velocity. Nevertheless, the requirements for computational storage and time are much higher than that of the numerical procedures based on the parabolic form such as those of Levy et al (1983) and Humphrey et al (1985) which, in turn, facilitate a much finer grid than has been possible when adopting the former method. The numerical studies of Chang et al (1983) and Choi et al (1989), in a square-sectioned U-bend, and Azzola et al (1986), in a circular cross-sectioned U-bend, indicated that grid independent solutions could be obtained with a grid density that was matched by the fine grid used in the present study. Although it could not be proved positively, with successive grid refinement, that the fine grid used here offered a grid independent solution, due to limitations in computational resources, it is evident that substantial grid refinements would be required to significantly improve the numerical accuracy of the fine grid prediction here.

The main objective of the investigation presented in this chapter is to provide a preliminary study of the flow inside a U-shaped curved exit nozzle of relevance to that of the reverse-flow annular combustor. With a higher inlet-to-exit area ratio in the exit nozzle, the flow would be expected to follow the same trends as in the model study of this chapter but higher maximum velocities are anticipated. However, the non-uniform inlet velocity profiles, which determine the influence of the secondary flow, would have a significant effect to the flow development inside the duct. The presence of cooling slots inside the curved nozzle would also alter the flow close to the two curved surfaces. A more thorough discussion of the flow in the curved exit nozzle is presented in Chapter 6.

4.6 Conclusions

This chapter presented experimental and numerical results of the flow inside a curved rectangular duct with radius-to-width ratio of 1.2 and aspect ratio of 3.8. The main findings are summarised in the following paragraphs:

1. The location of the maximum streamwise velocity remained close to the inside surface of the bend with maximum of $1.3 U_b$ due to the strong radial pressure gradient and this led to strong radial velocity towards the inside surface with maximum of $0.25 U_b$.
2. The influence of secondary flow, with magnitude of about $0.1 U_b$, was limited in its extent in the spanwise direction to about one third of the width of the duct due to the high aspect ratio.

3. The flow development in the curved nozzle is independent to the inlet Reynolds number but is more sensitive to the inlet velocity profile which determines the strength of the secondary velocity.
4. A two-equation eddy viscosity model is able to predict the flow development within a curved duct where pressure gradient was the more influential factor whereas for regions where secondary flow dominated the flow pattern, the numerical procedures were unable to predict the flow characteristics due to the assumption of local isotropy.

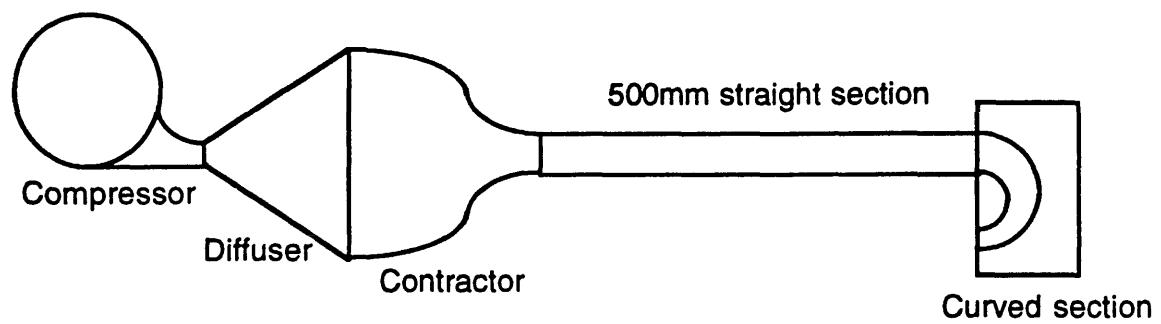
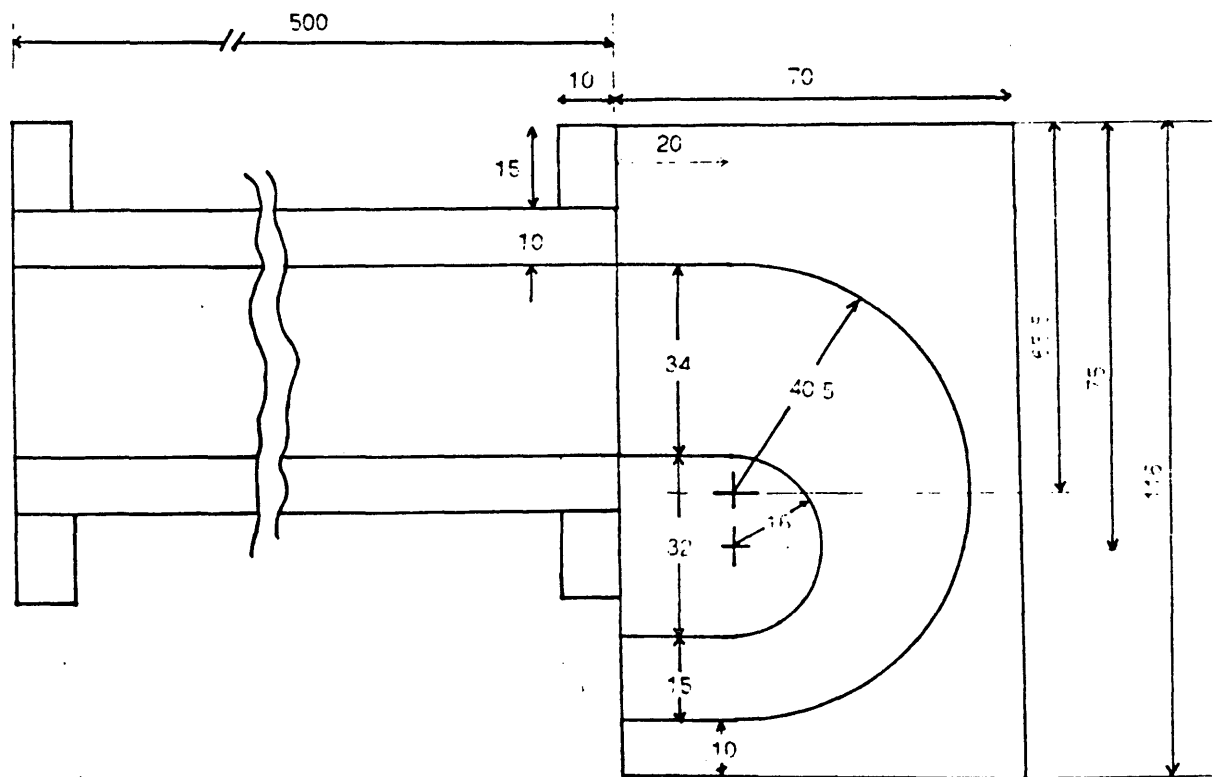


Figure 4.1 Flow Arrangement



All Dimensions in mm

Figure 4.2 Geometry of Curved Nozzle

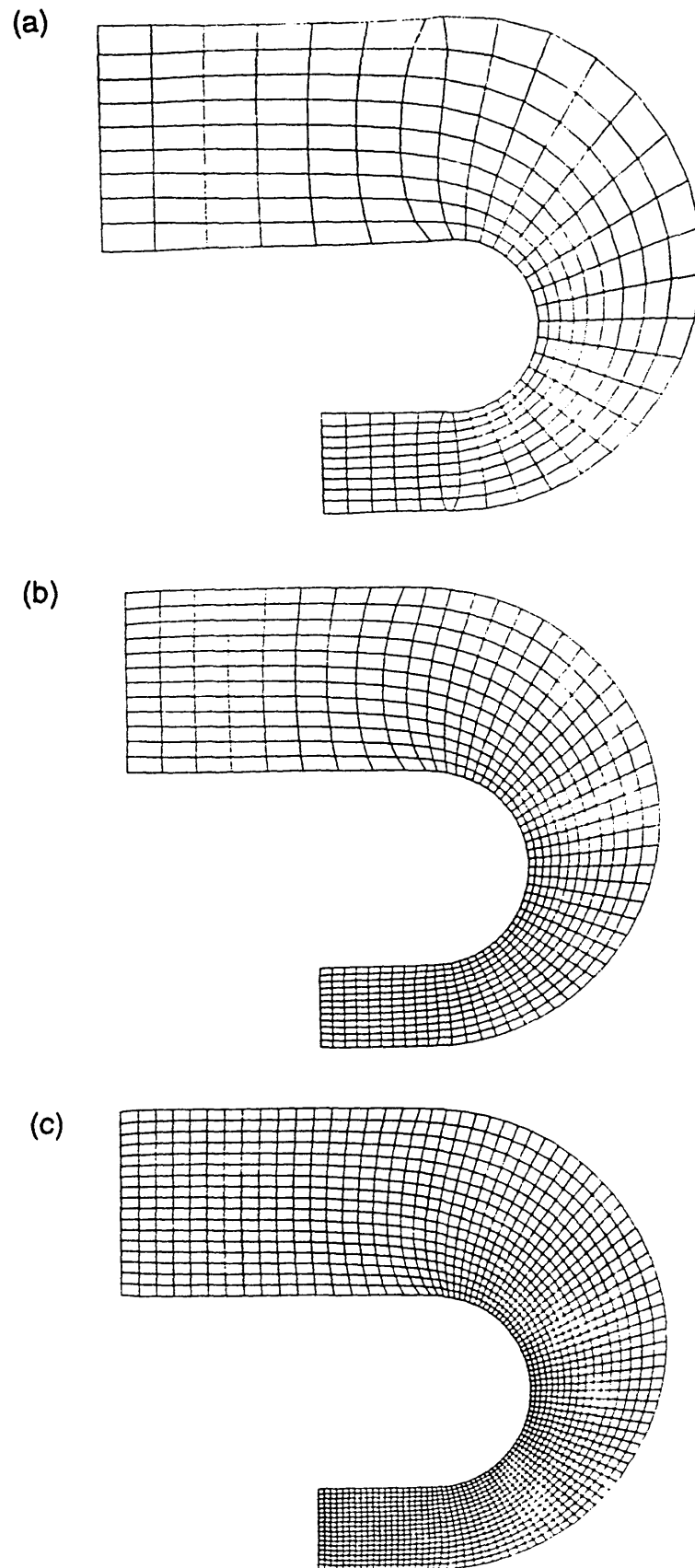
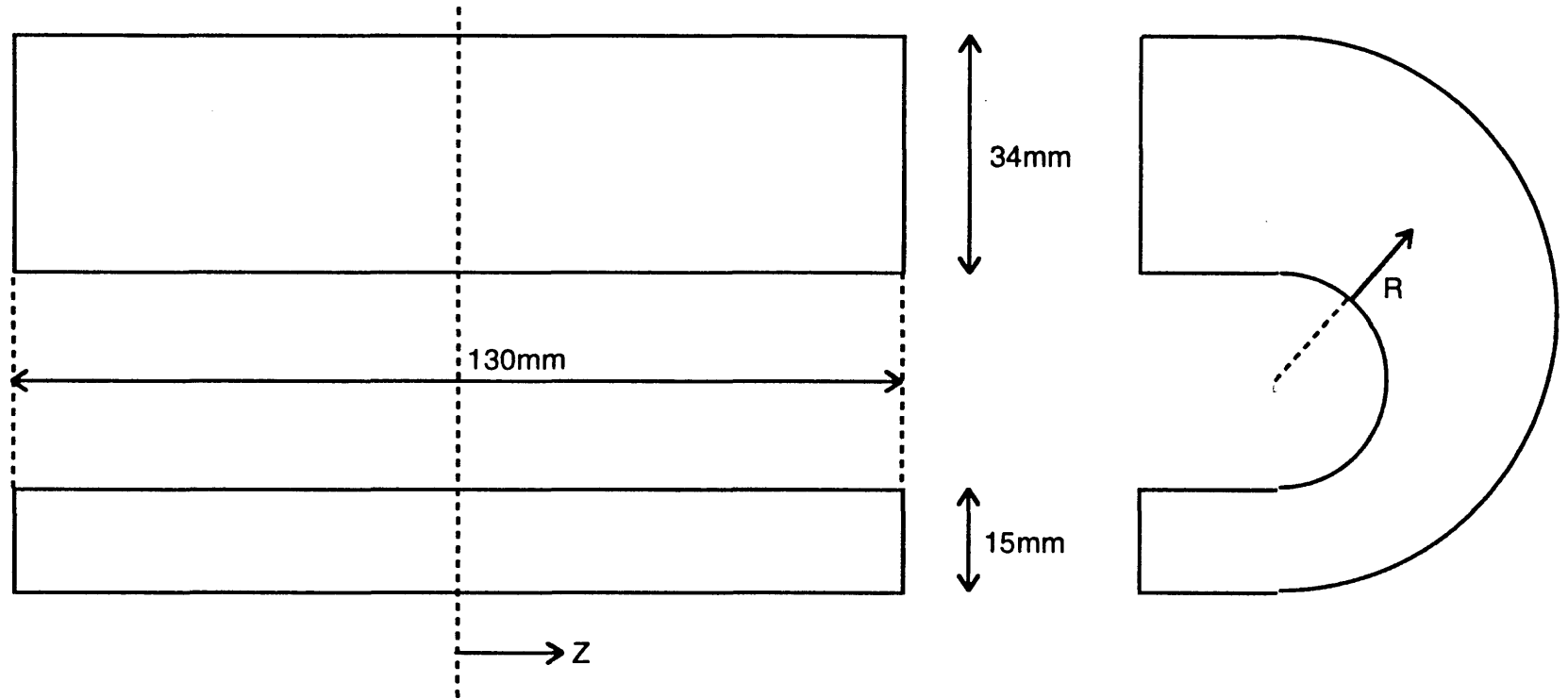


Figure 4.3 Finite Difference Meshes used in the Calculations
(a) Coarse Grid (32x10x25 nodes)
(b) Standard Grid (58x15x25 nodes)
(c) Fine Grid (101x20x35 nodes)



$$Z^* = Z / 65 \text{ mm}$$

$$R^* = R / (\Delta R) \quad \text{where } \Delta R \text{ is the local duct width}$$

Figure 4.4 Co-ordinates System used for the Curved Nozzle

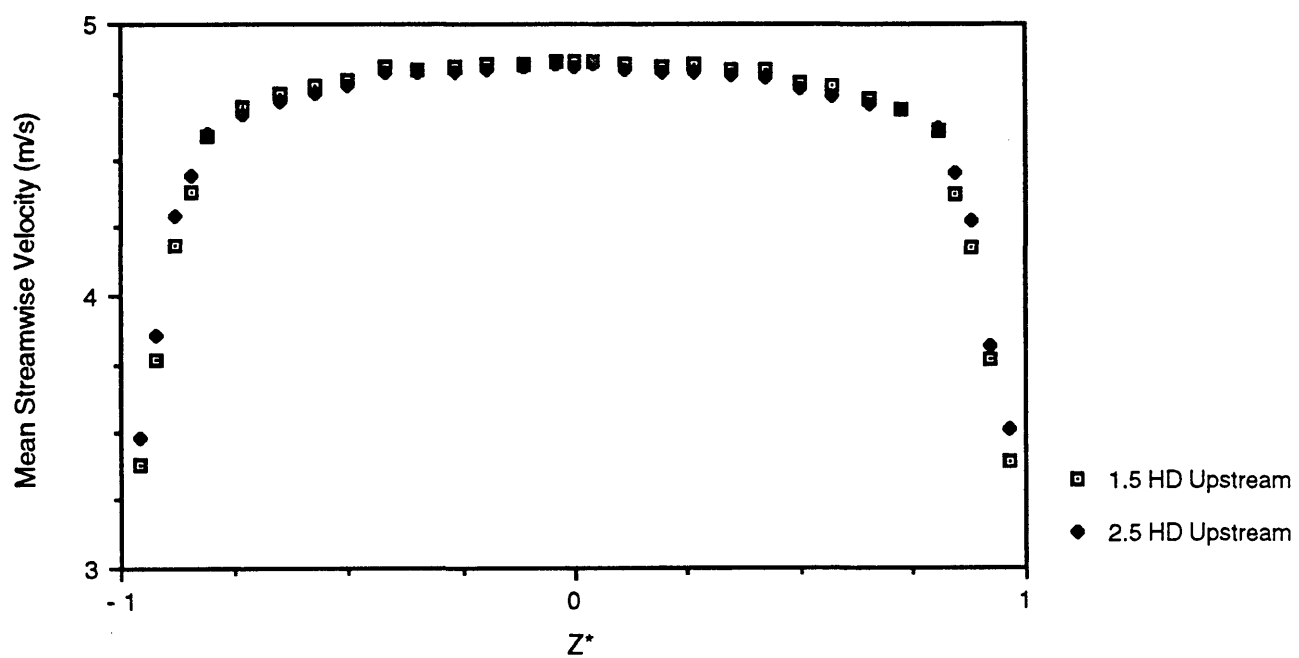


Figure 4.5 Measurements of Mean Streamwise Velocities Upstream of Entry to Curved Nozzle for $Re_{inlet}=15,000$

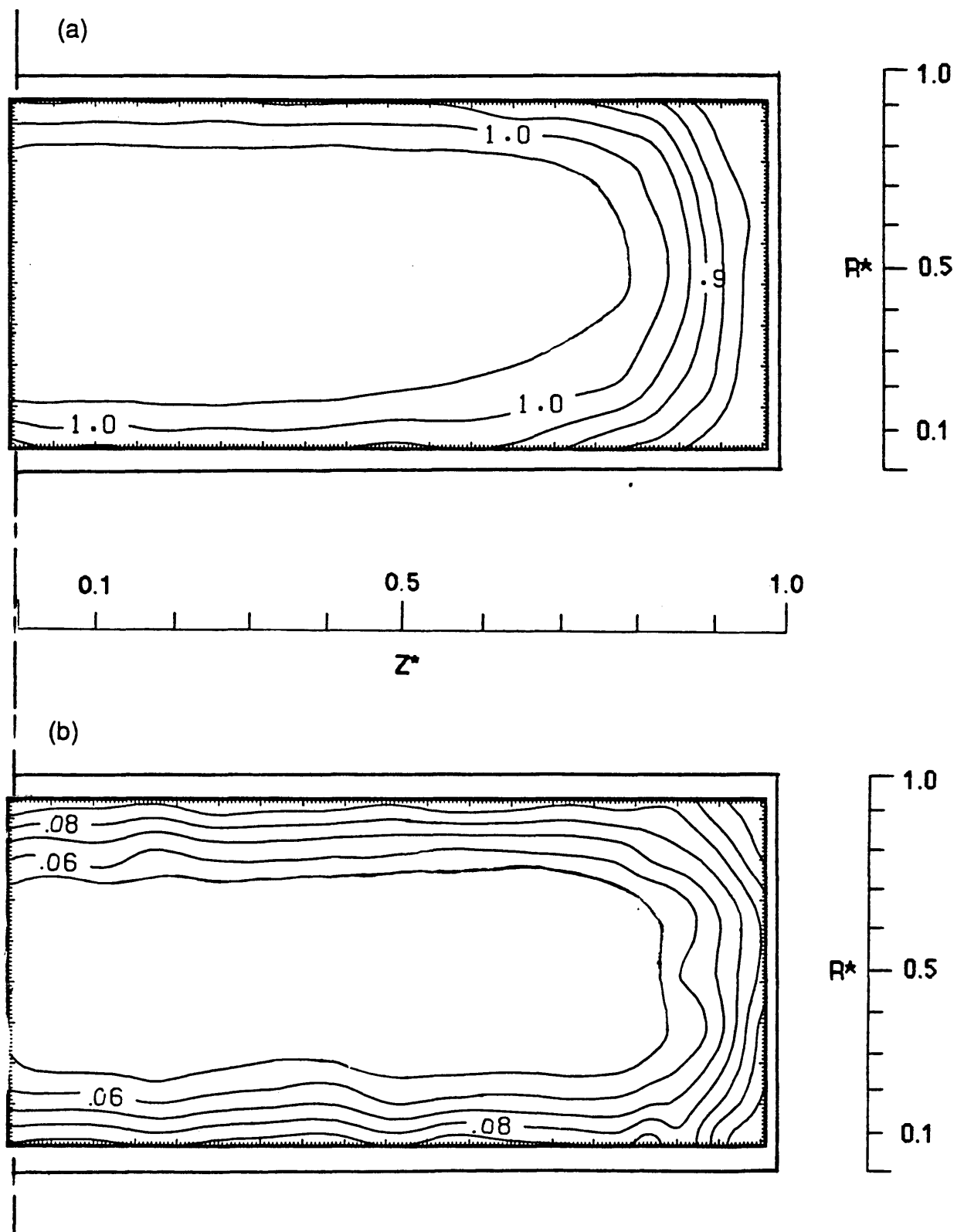


Figure 4.6 Measurements of Mean and rms Streamwise Velocity at 1 Hydraulic Diameter Upstream of Entry to Curved Nozzle for $Re_{inlet}=15,000$

(a) Mean (b) rms

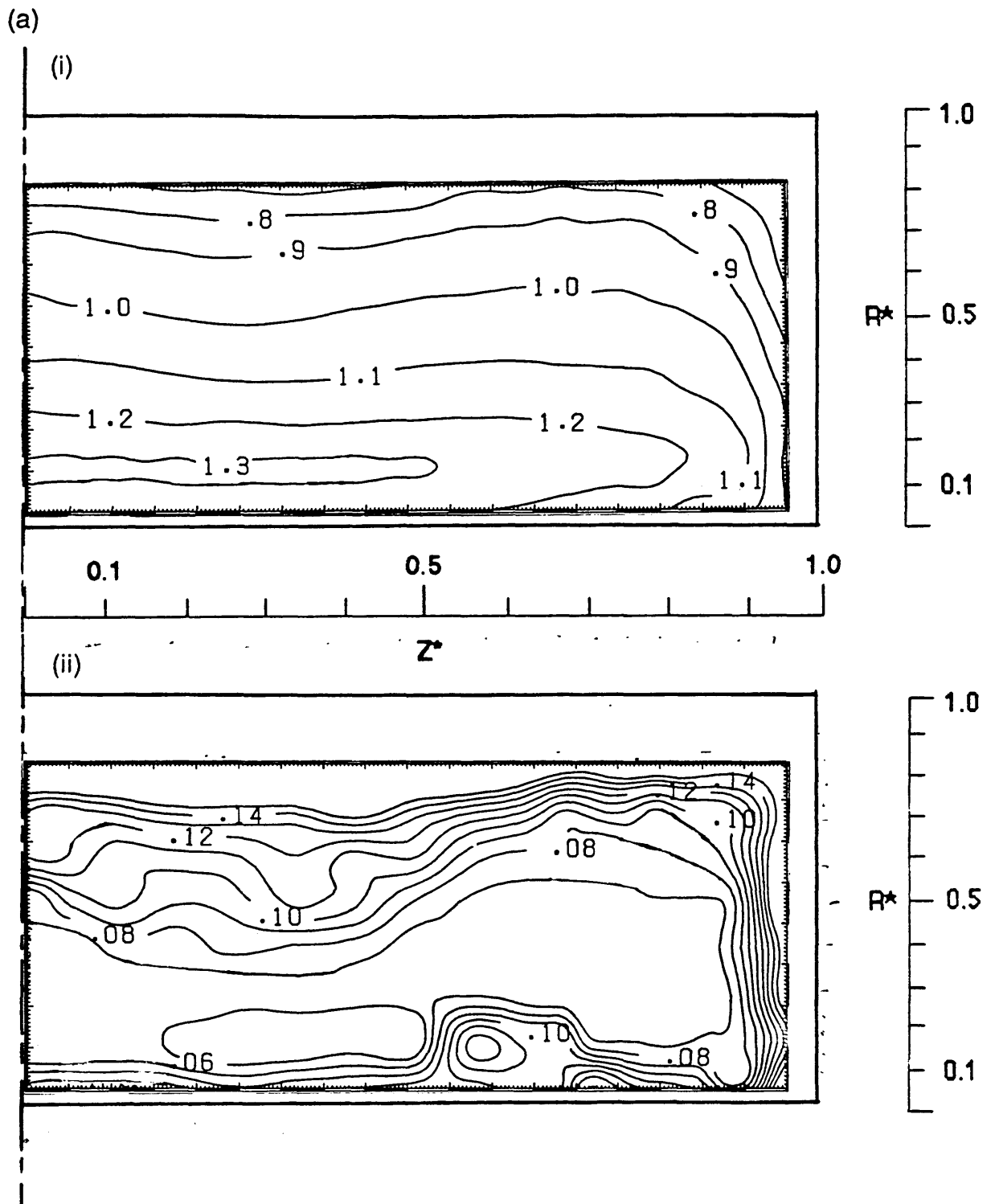


Figure 4.7 Measurements of Mean and rms Velocity at Entry to Nozzle (0°

Plane) for $Re_{inlet}=15,000$

(a) Streamwise Velocity

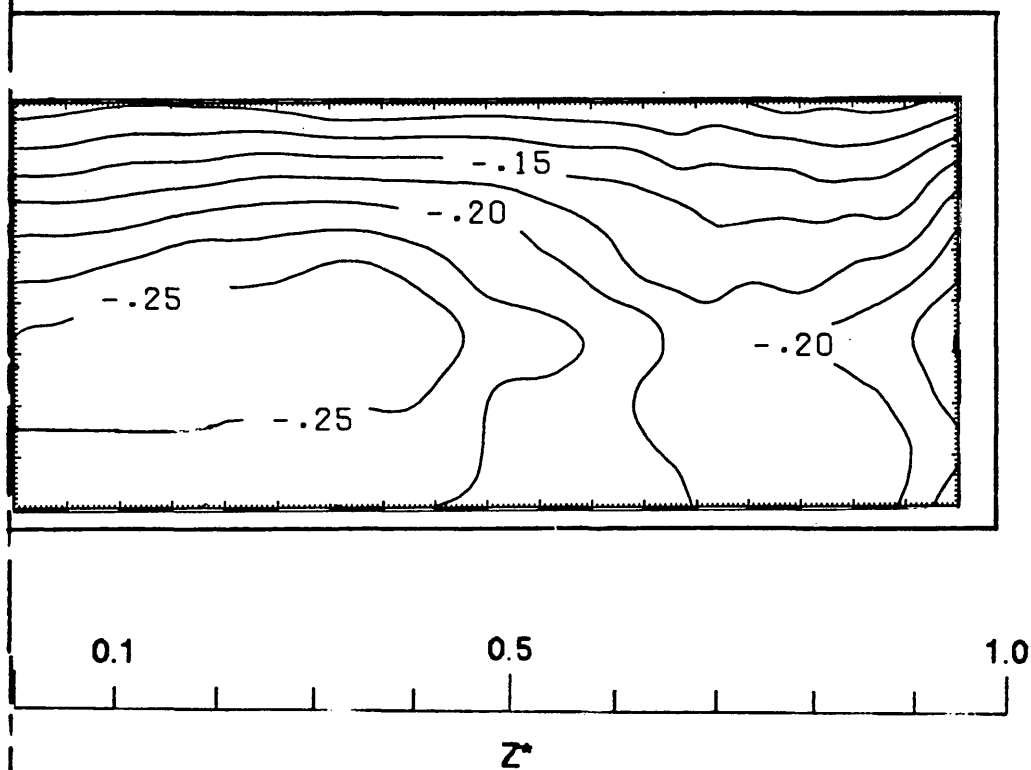
(i) Mean (ii) rms

(b) Radial Velocity

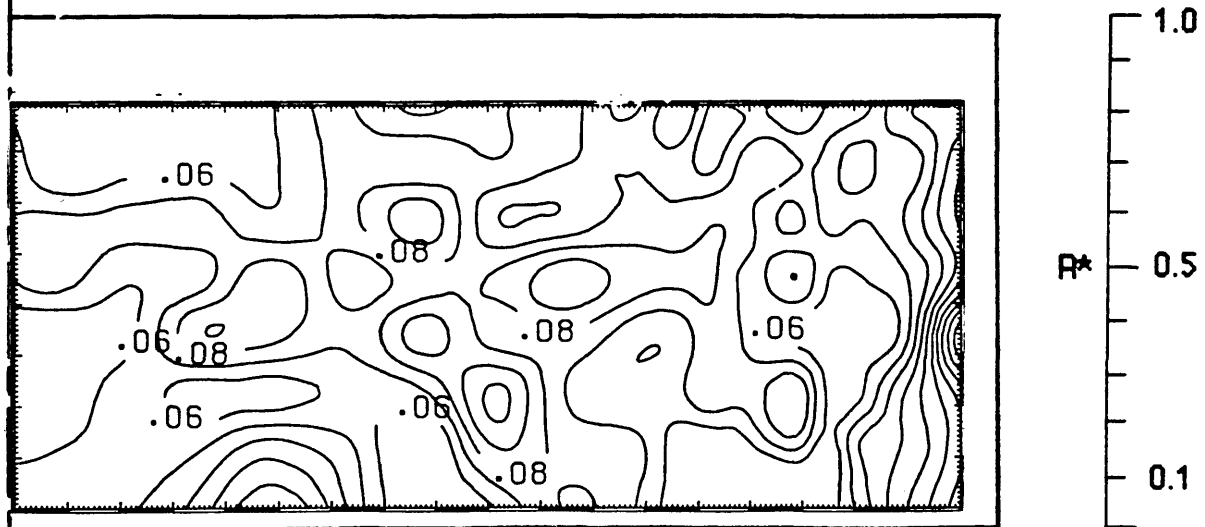
(i) Mean (ii) rms

(b)

(i)



(ii)



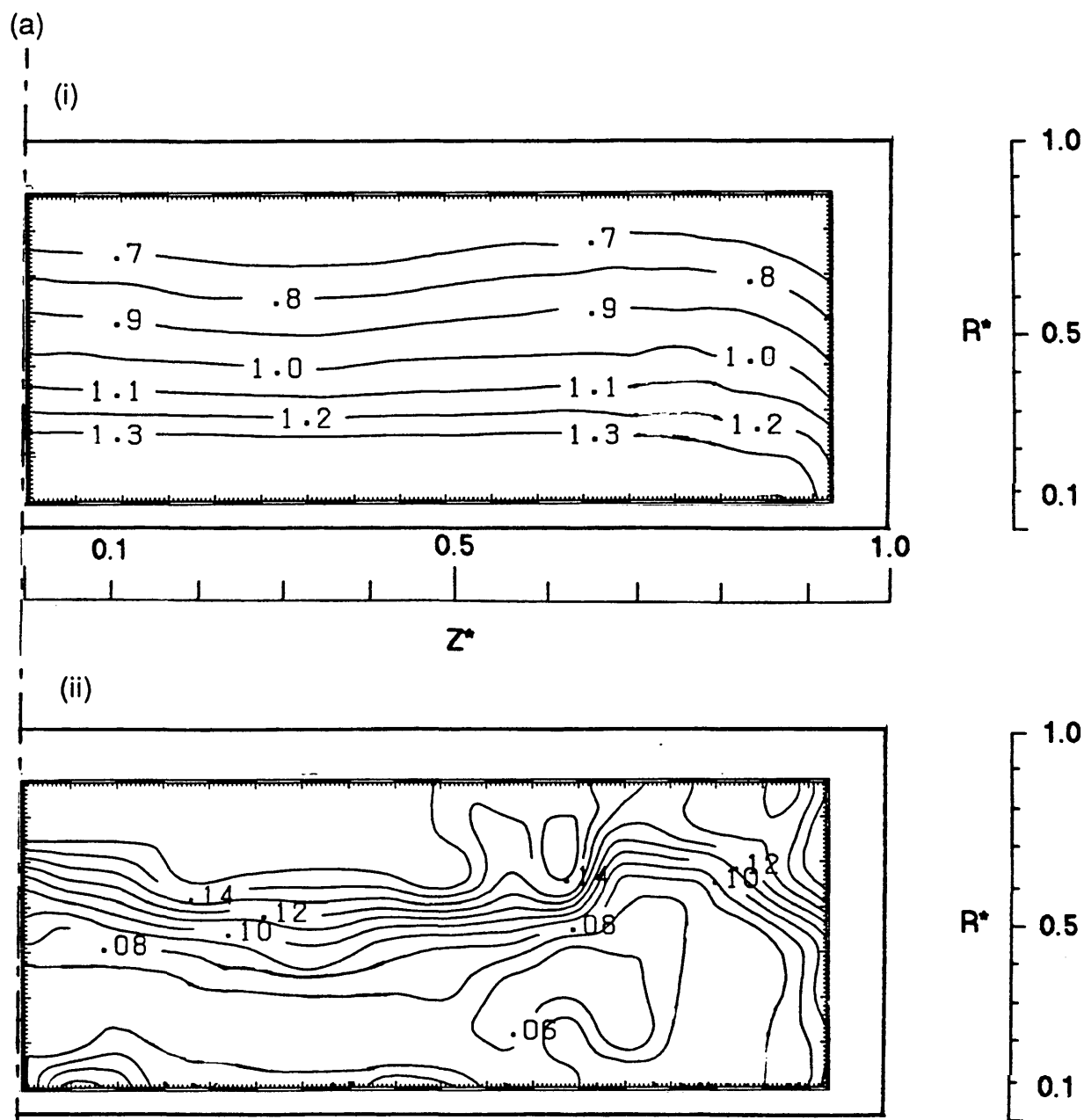
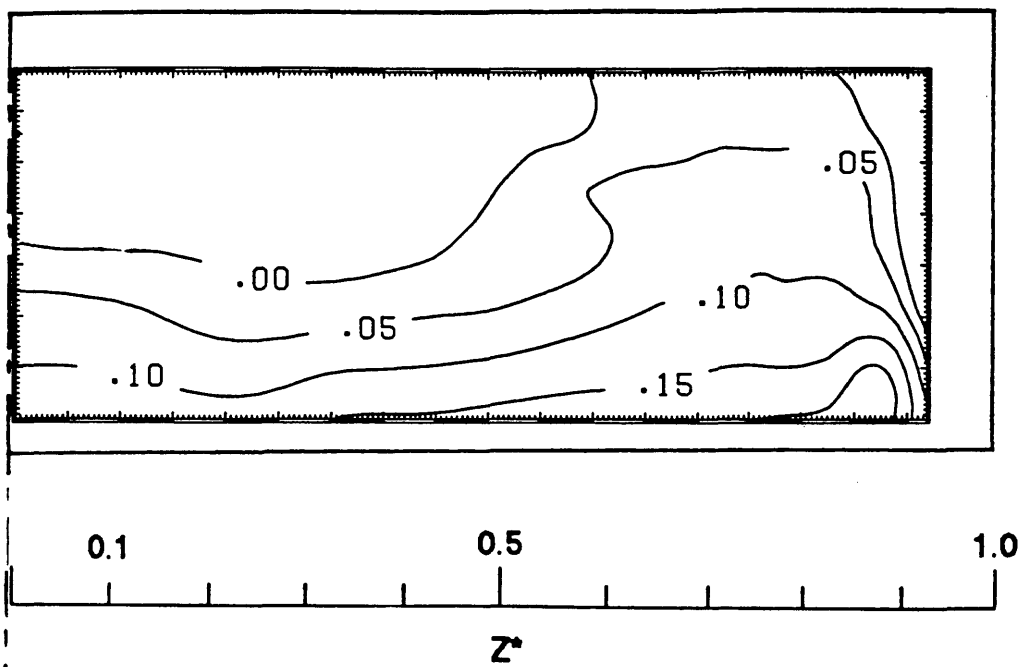


Figure 4.8 Measurements of Mean and rms Velocity at 45° Plane for $Re_{inlet}=15,000$

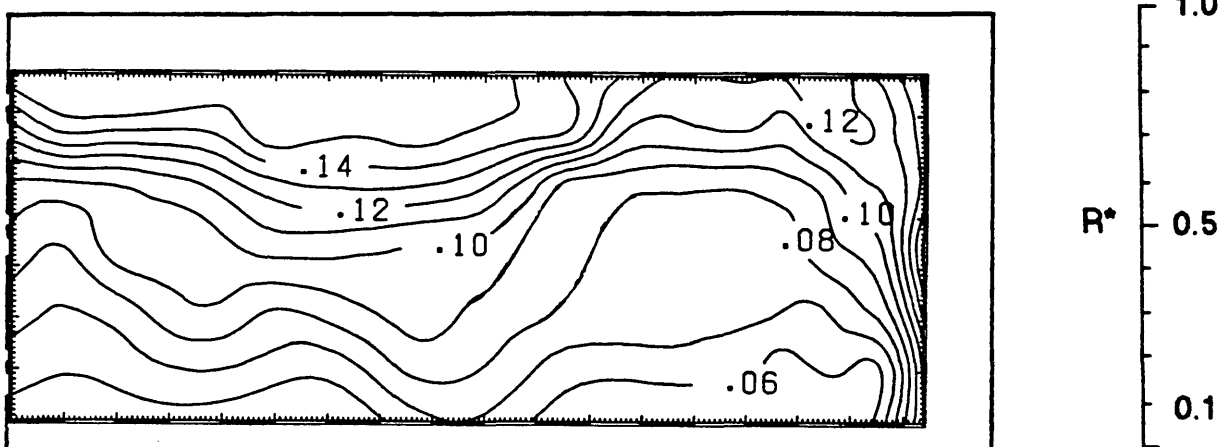
- (a) Streamwise Velocity
 (i) Mean (ii) rms
 (b) Radial Velocity
 (i) Mean (ii) rms

(b)

(i)



(ii)



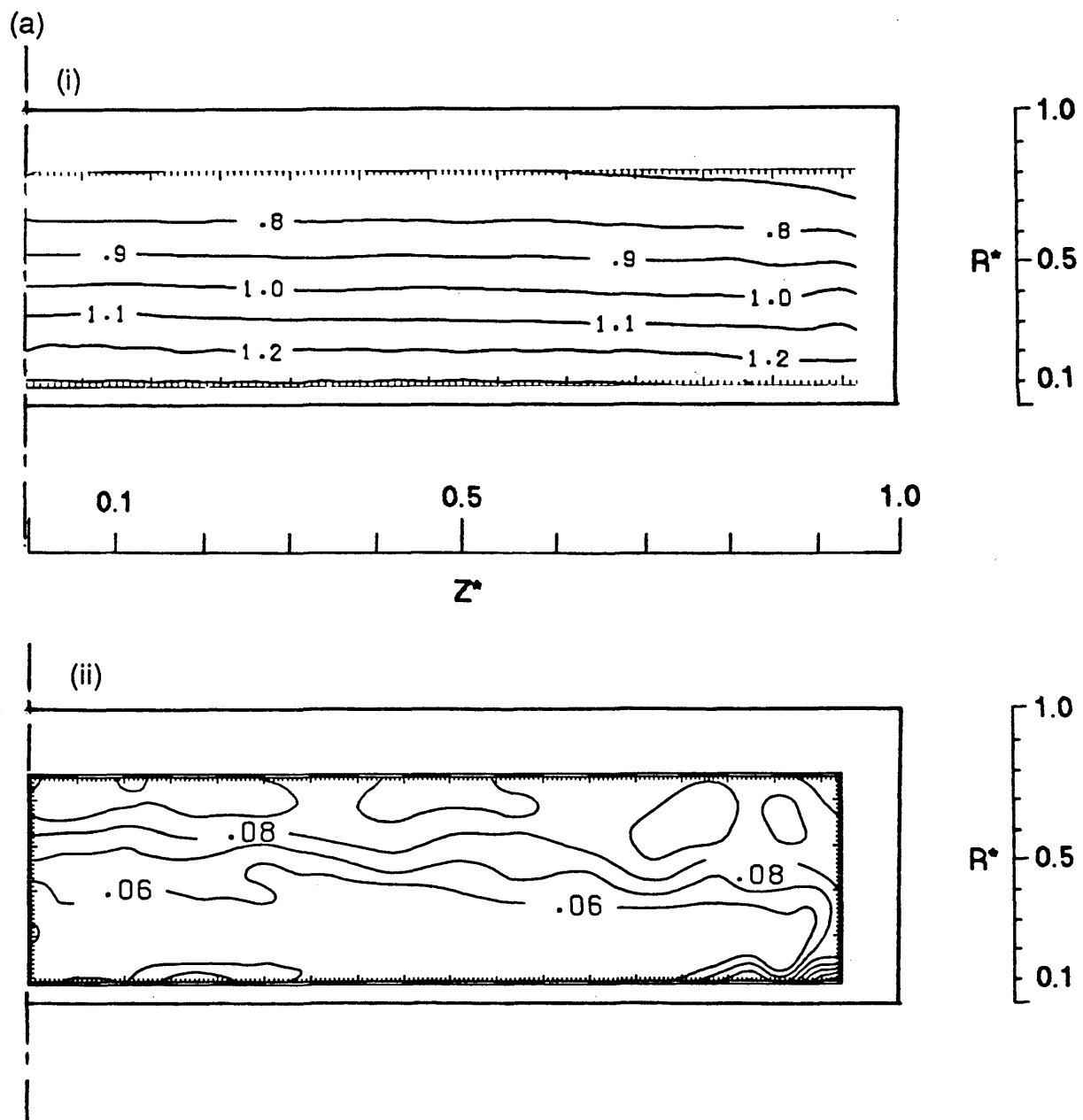


Figure 4.9 Measurements of Mean and rms Velocity at 90° Plane for

$Re_{inlet}=15,000$

(a) Streamwise Velocity

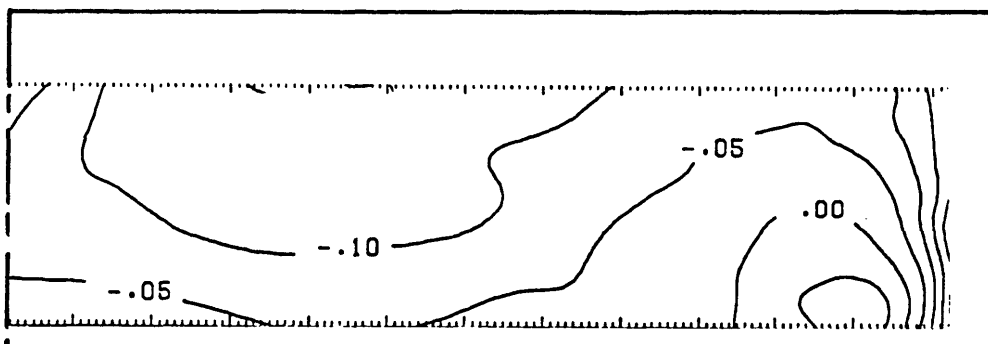
(i) Mean (ii) rms

(b) Radial Velocity

(i) Mean (ii) rms

(b)

(i)



R^*

1.0
0.5
0.1

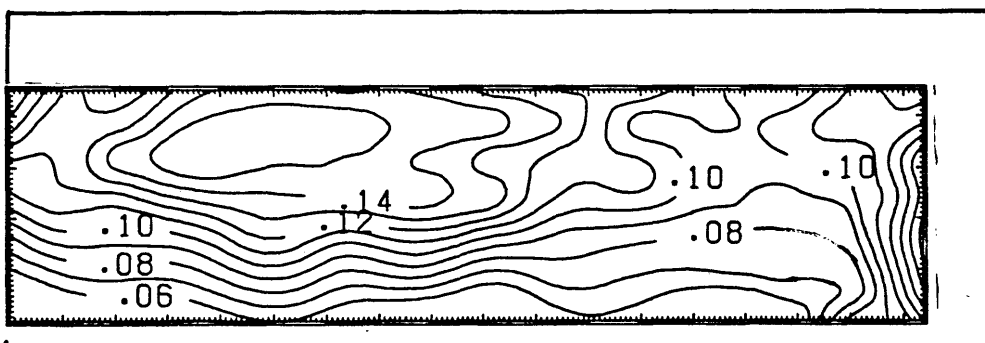
0.1

0.5

1.0

Z^*

(ii)



R^*

1.0
0.5
0.1

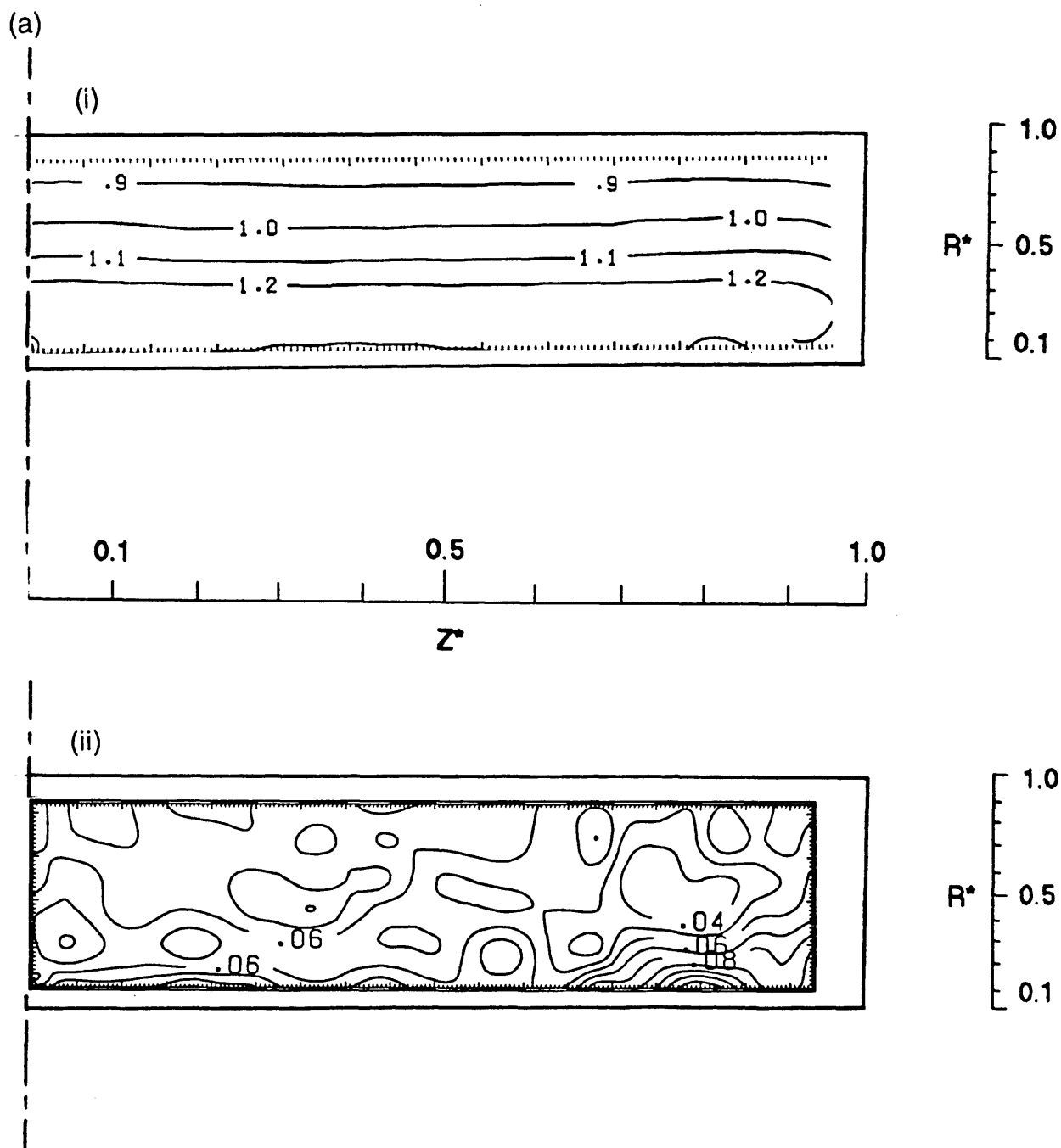


Figure 4.10 Measurements of Mean and rms Velocity at 135° Plane for

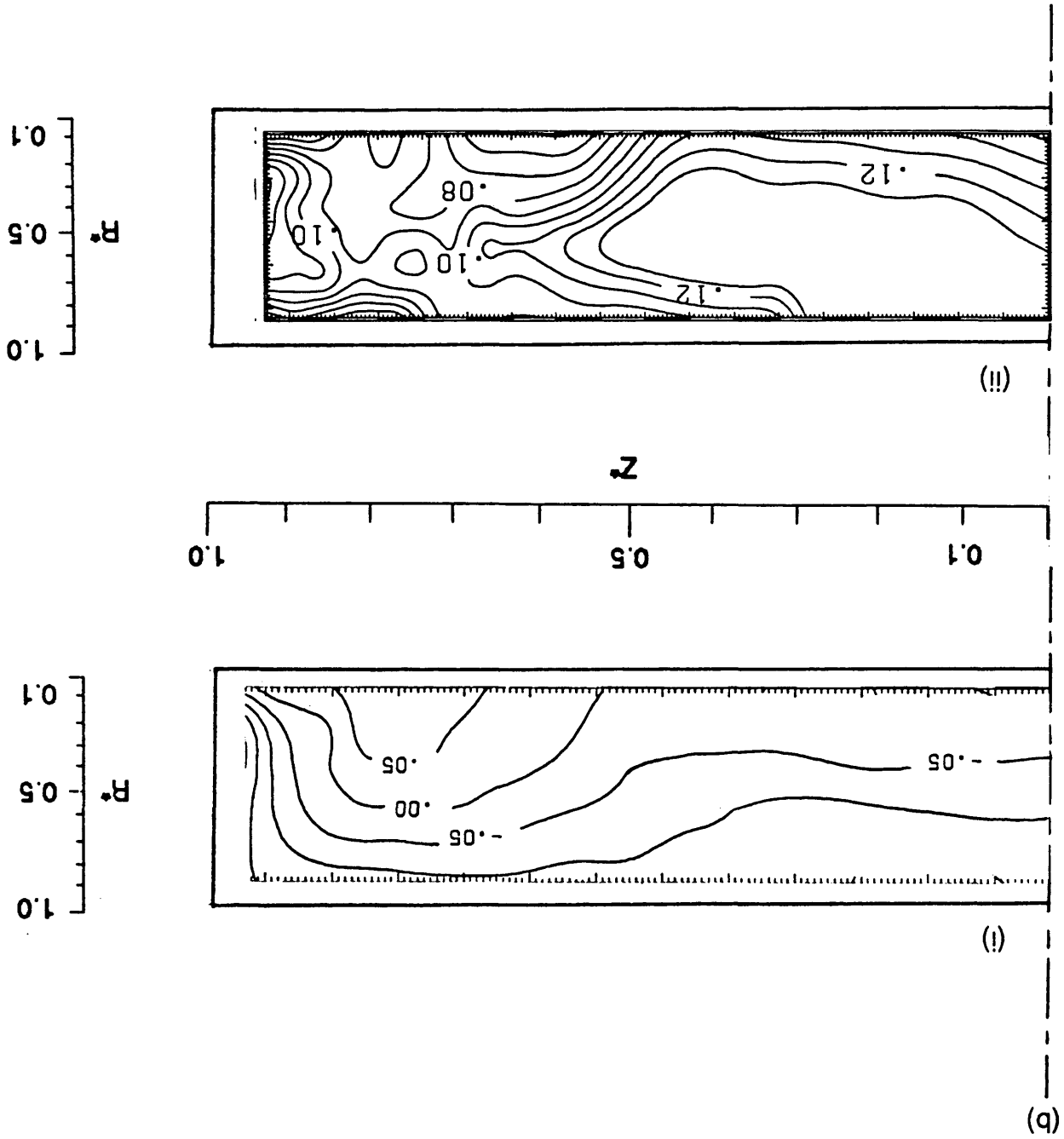
$Re_{inlet}=15,000$

(a) Streamwise Velocity

(i) Mean (ii) rms

(b) Radial Velocity

(i) Mean (ii) rms



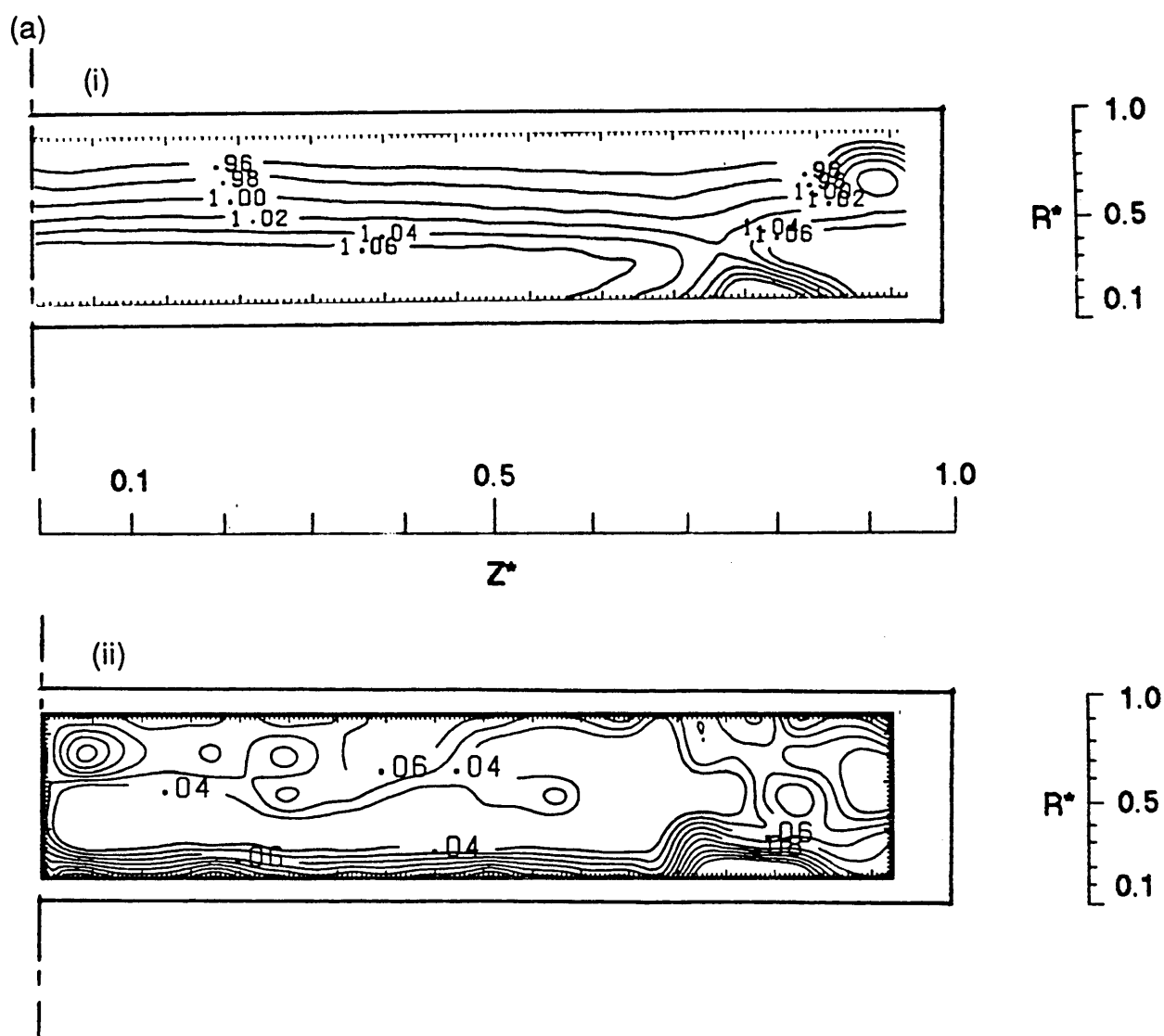


Figure 4.11 Measurements of Mean and rms Velocity at 180° Plane for

$Re_{inlet}=15,000$

(a) Streamwise Velocity

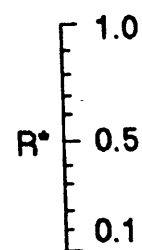
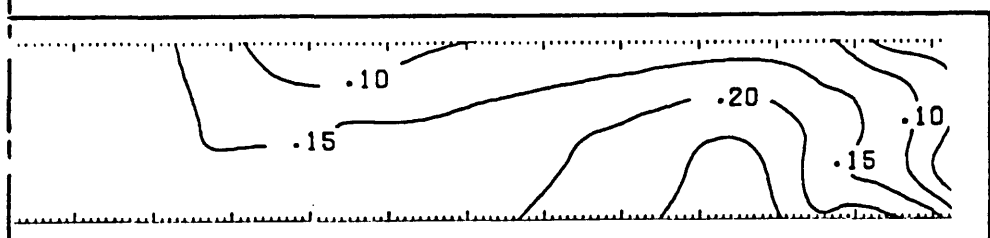
(i) Mean (ii) rms

(b) Radial Velocity

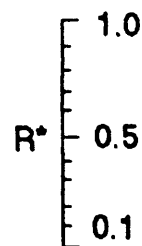
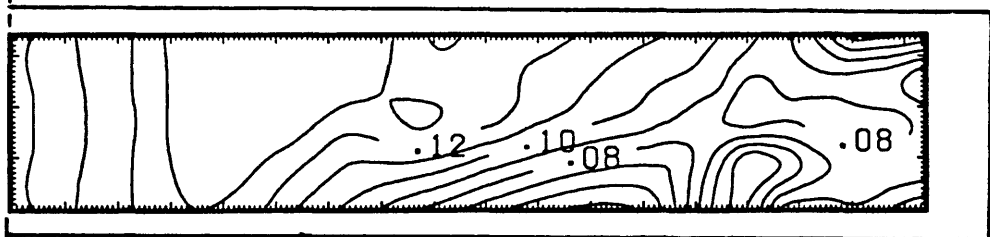
(i) Mean (ii) rms

(b)

(i)



(ii)



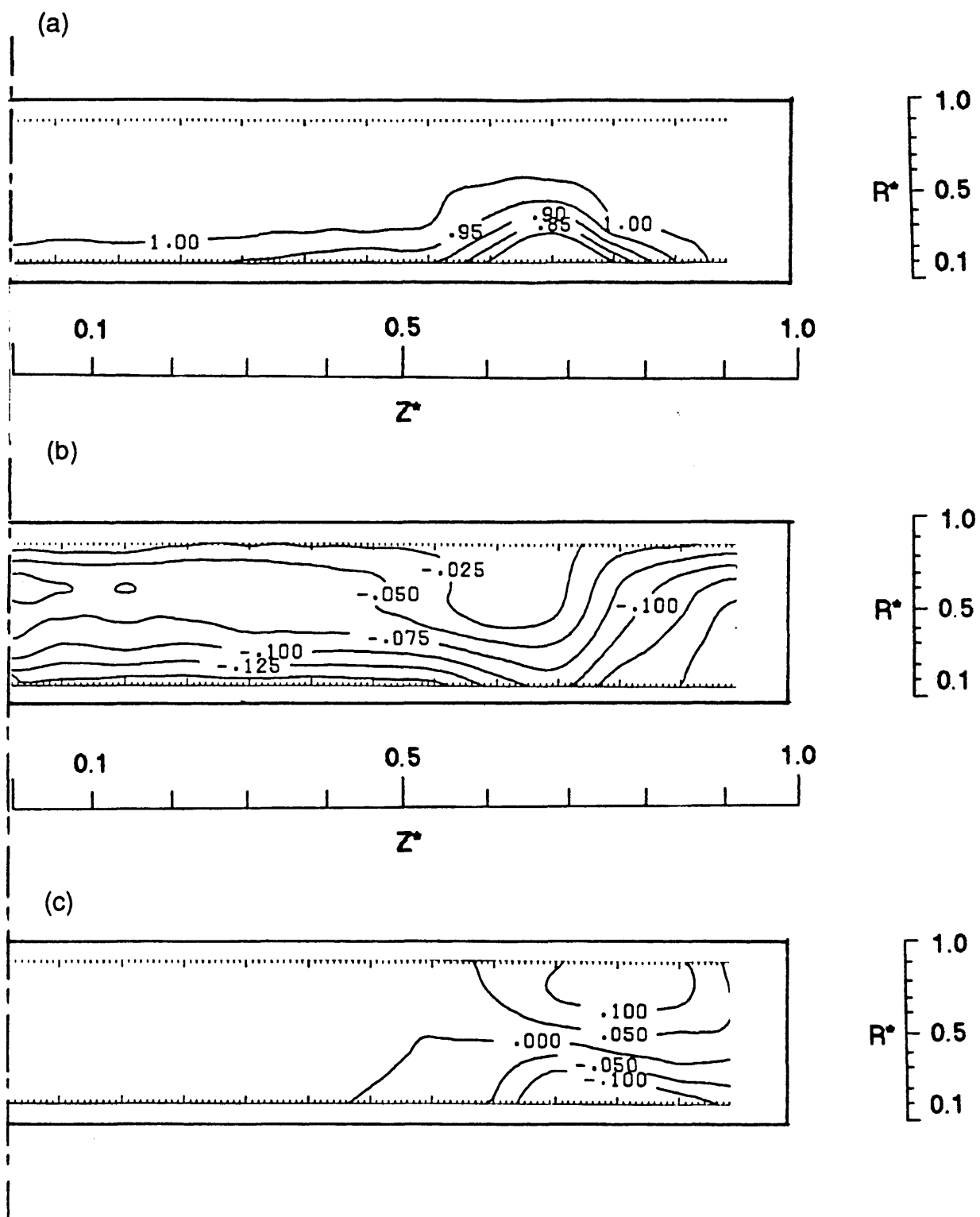


Figure 4.12 Measurements of Mean Velocity at Exit of Nozzle for

$Re_{inlet}=15,000$

(a) Streamwise Velocity (b) Radial Velocity (c) W-Velocity

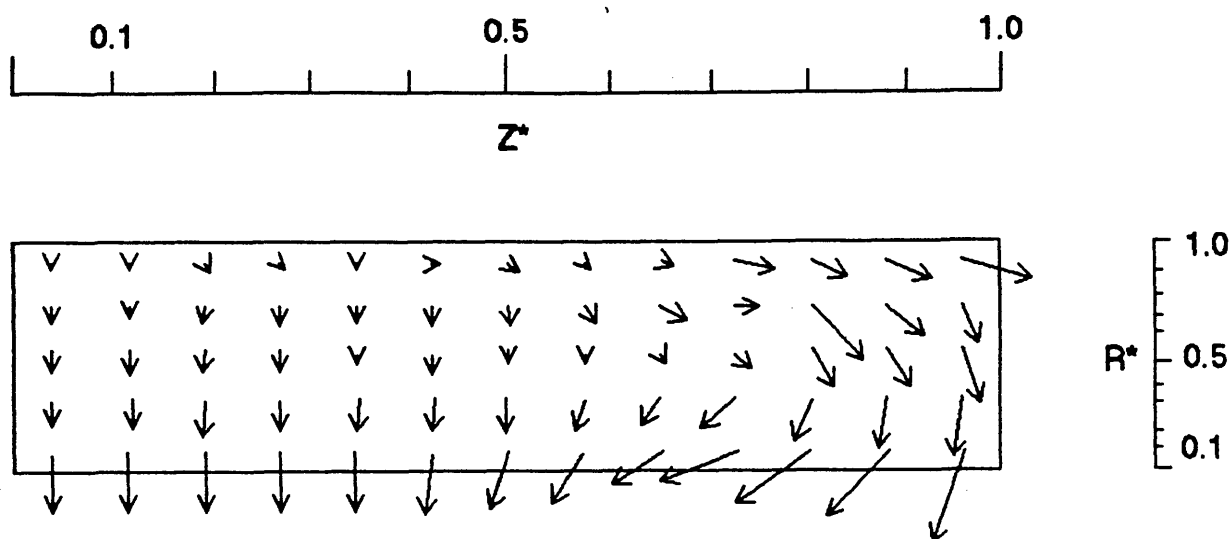


Figure 4.13 Vector Plot of Measured Velocity at Exit of Nozzle for $Re_{inlet}=15,000$

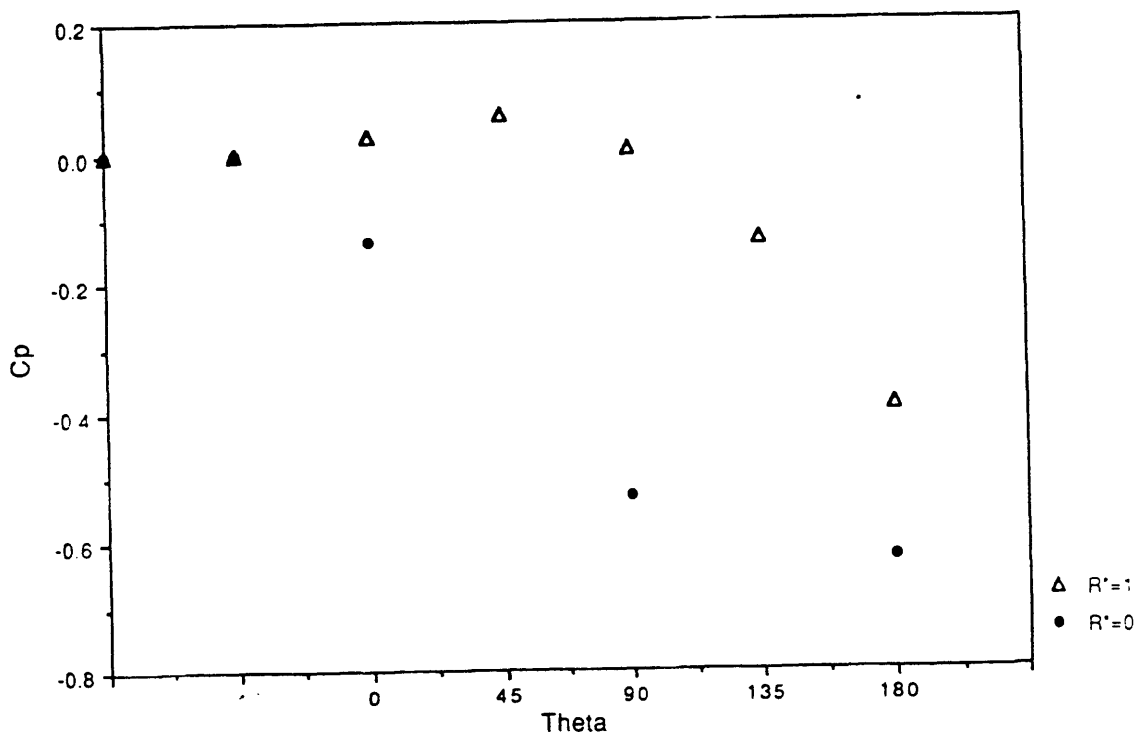


Figure 4.14 Measured Pressure Distribution at $Z^*=0$ for $Re_{inlet}=15,000$

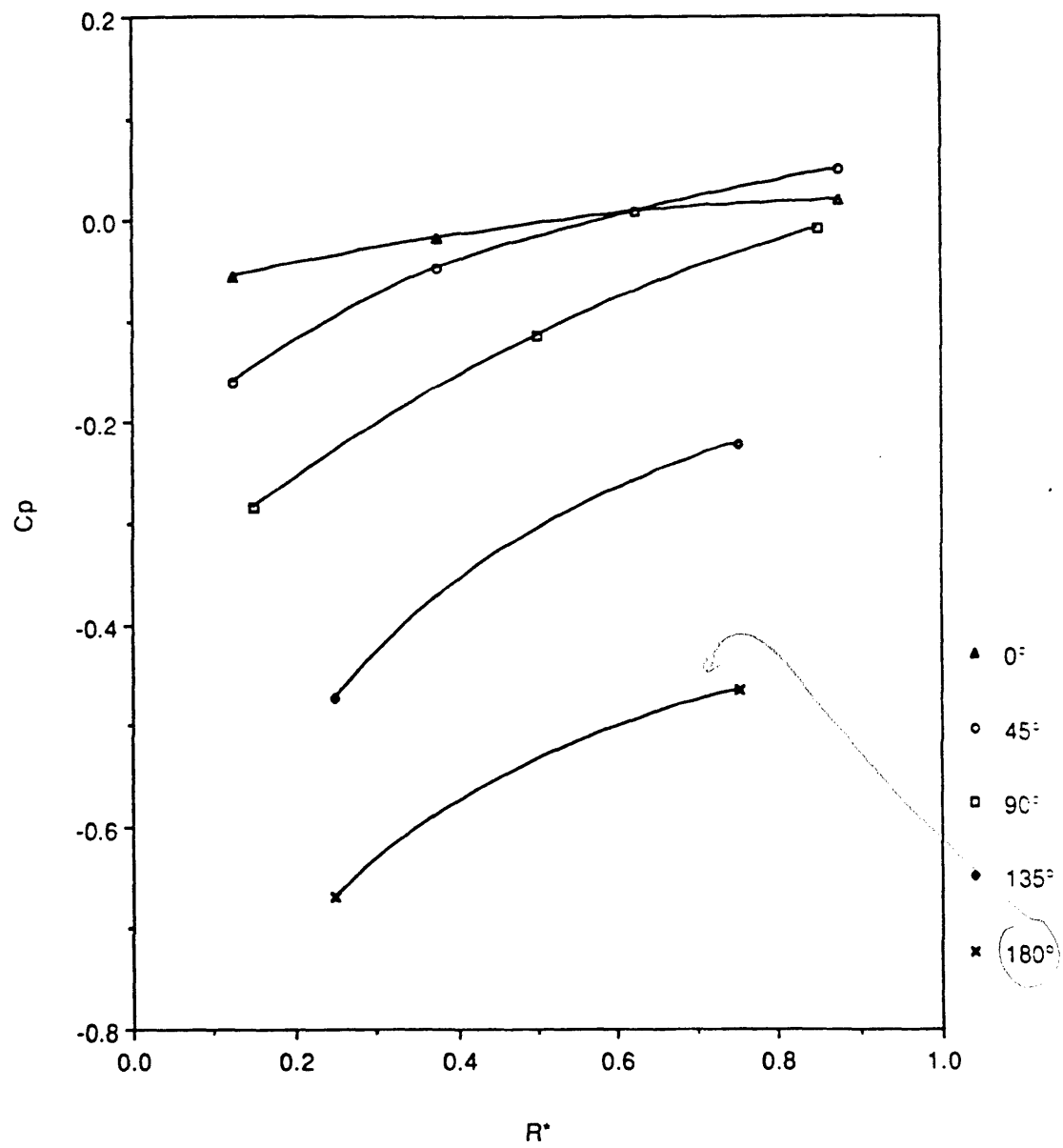


Figure 4.15 Measured Pressure Distribution at $Z^*=1$ for $Re_{inlet}=15,000$

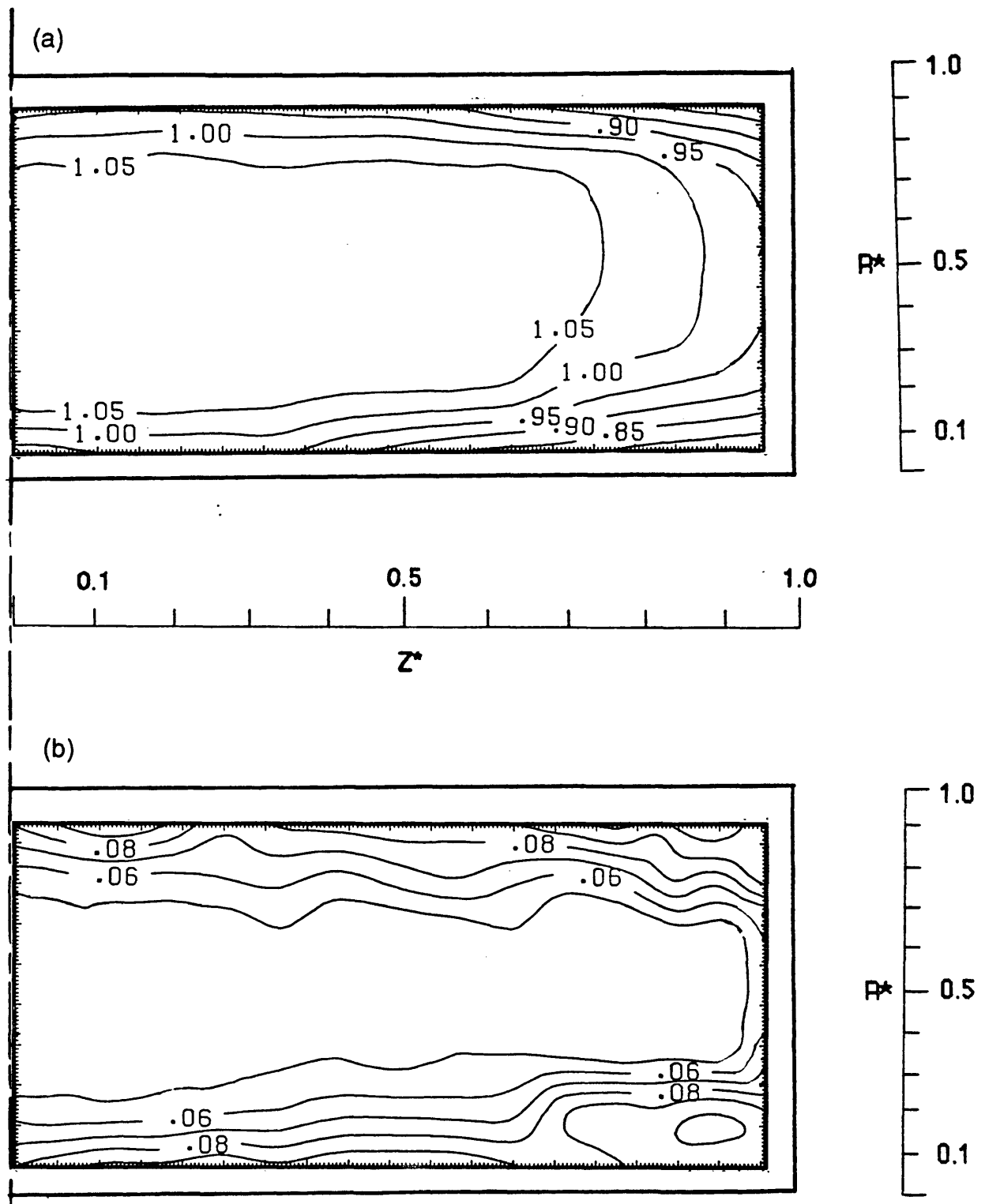


Figure 4.16 Measurements of Mean and rms Streamwise Velocity at 1 Hydraulic Diameter Upstream of Entry to Curved Nozzle for $Re_{inlet}=40,000$

(a) Mean

(b) rms

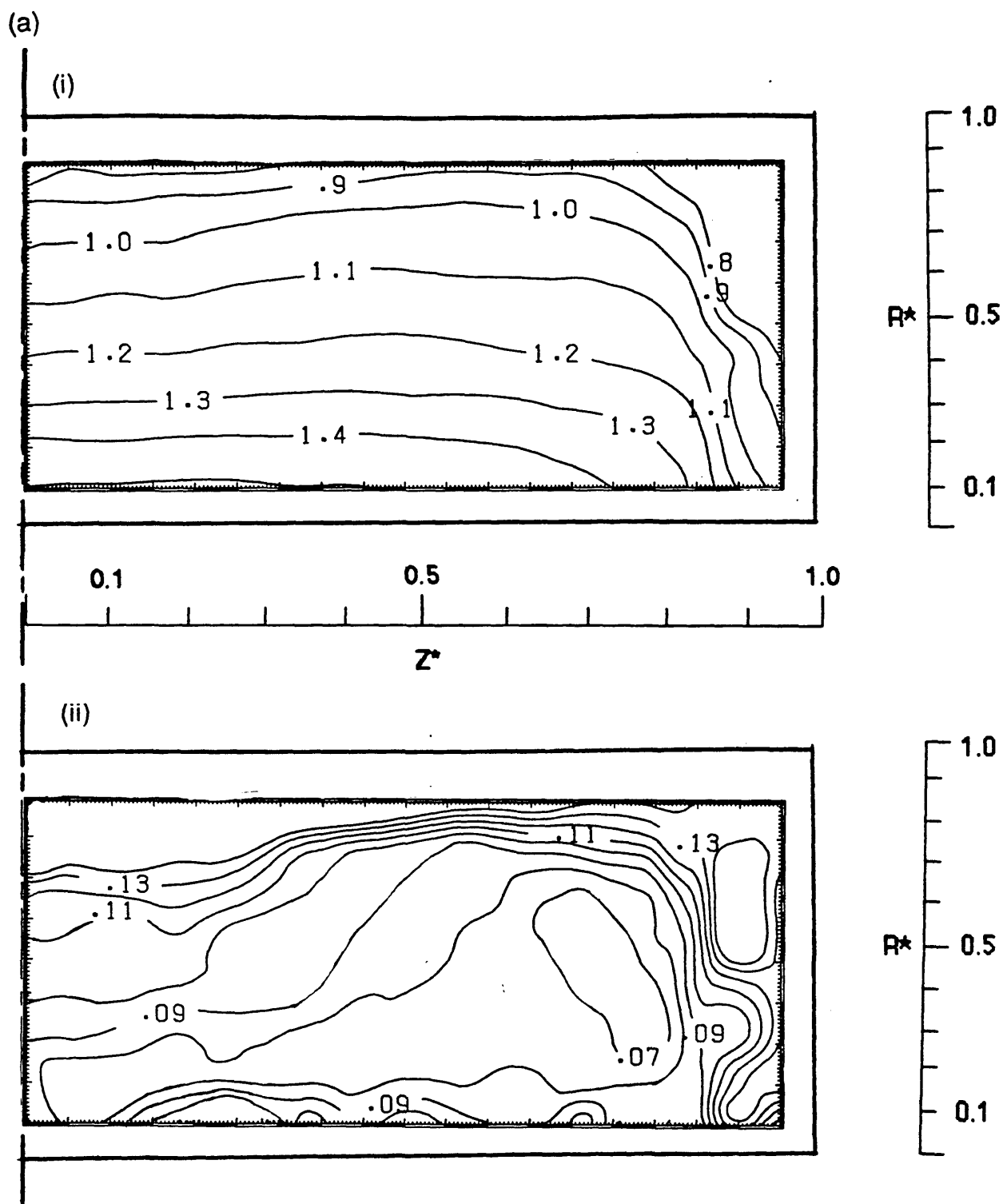


Figure 4.17 Measurements of Mean and rms Velocity at Entry to Nozzle (0°

Plane) for $Re_{inlet}=40,000$

(a) Streamwise Velocity

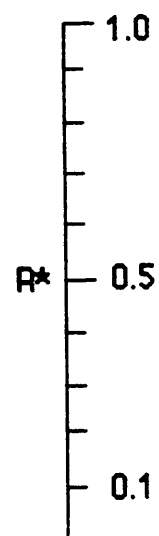
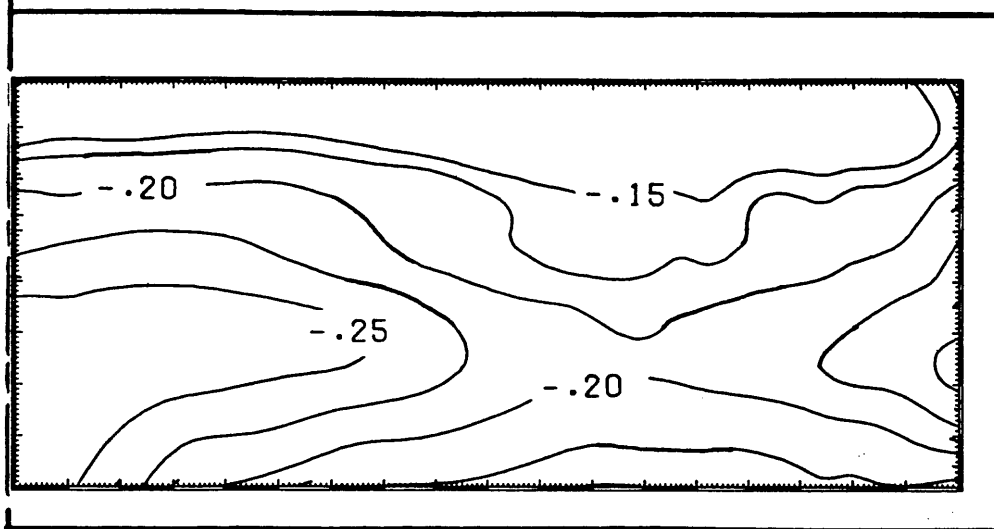
(i) Mean (ii) rms

(b) Radial Velocity

(i) Mean (ii) rms

(b)

(i)



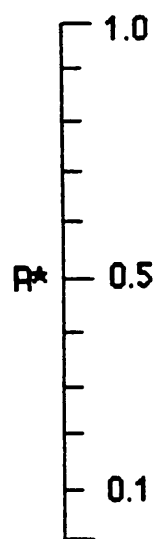
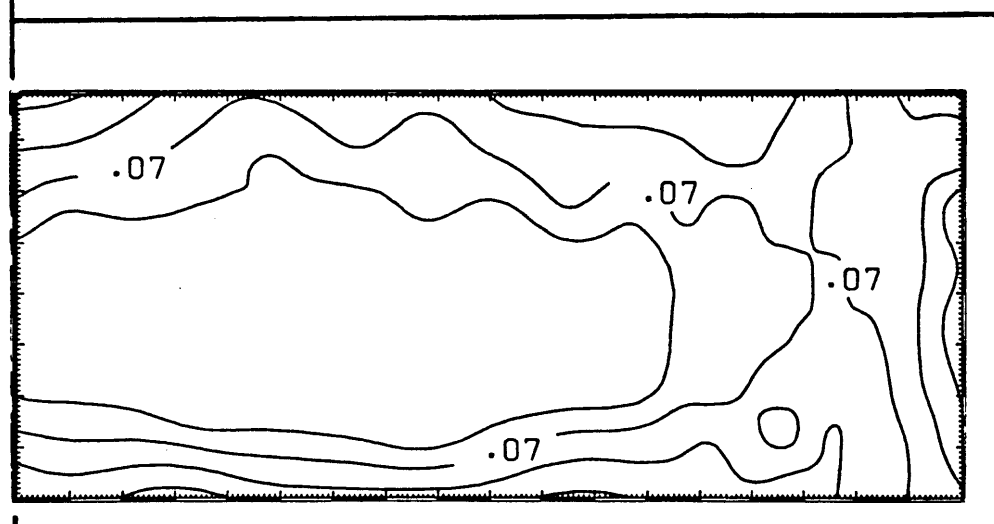
0.1

0.5

1.0

Z^*

(ii)



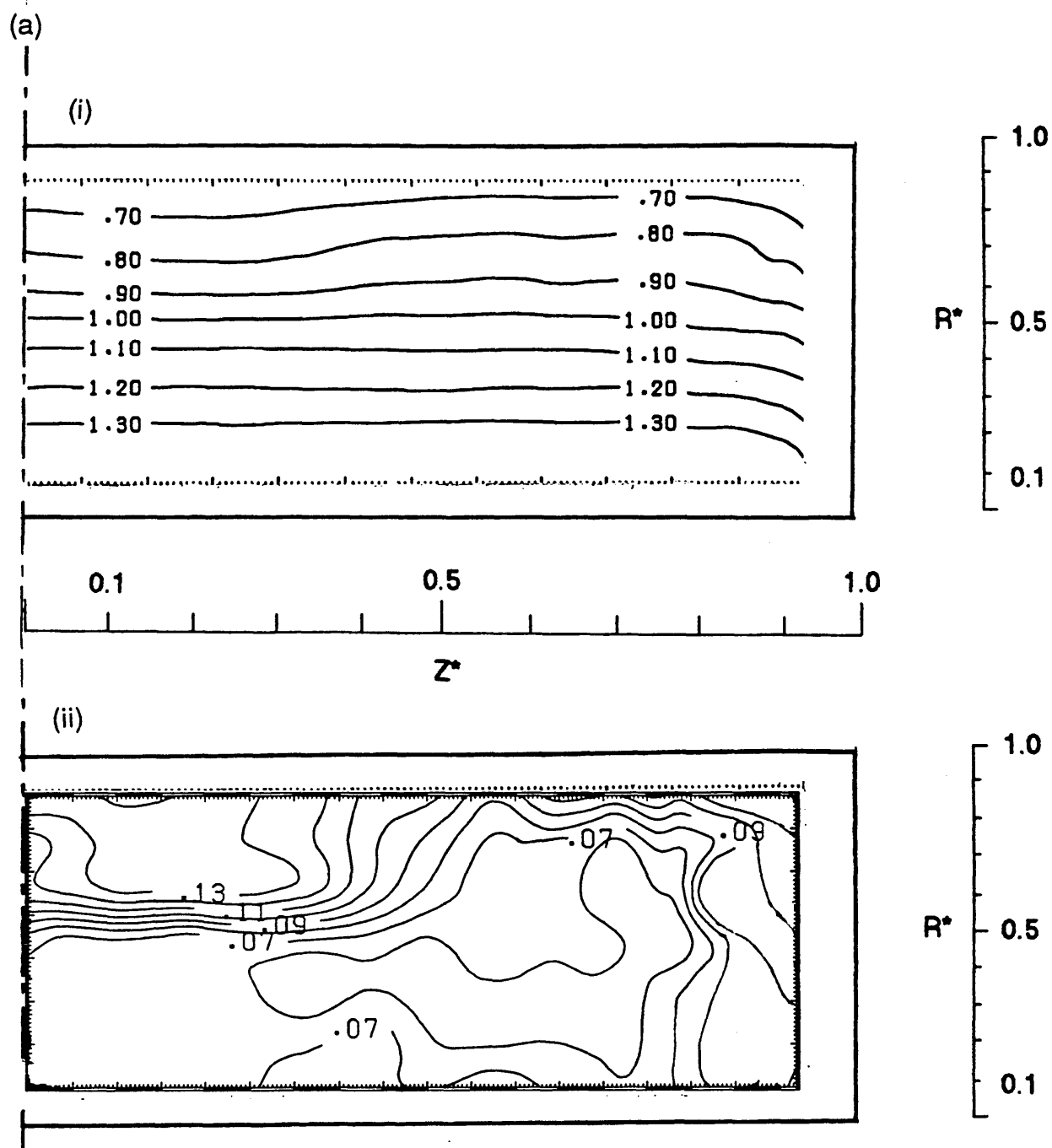


Figure 4.18 Measurements of Mean and rms Velocity at 45° Plane for

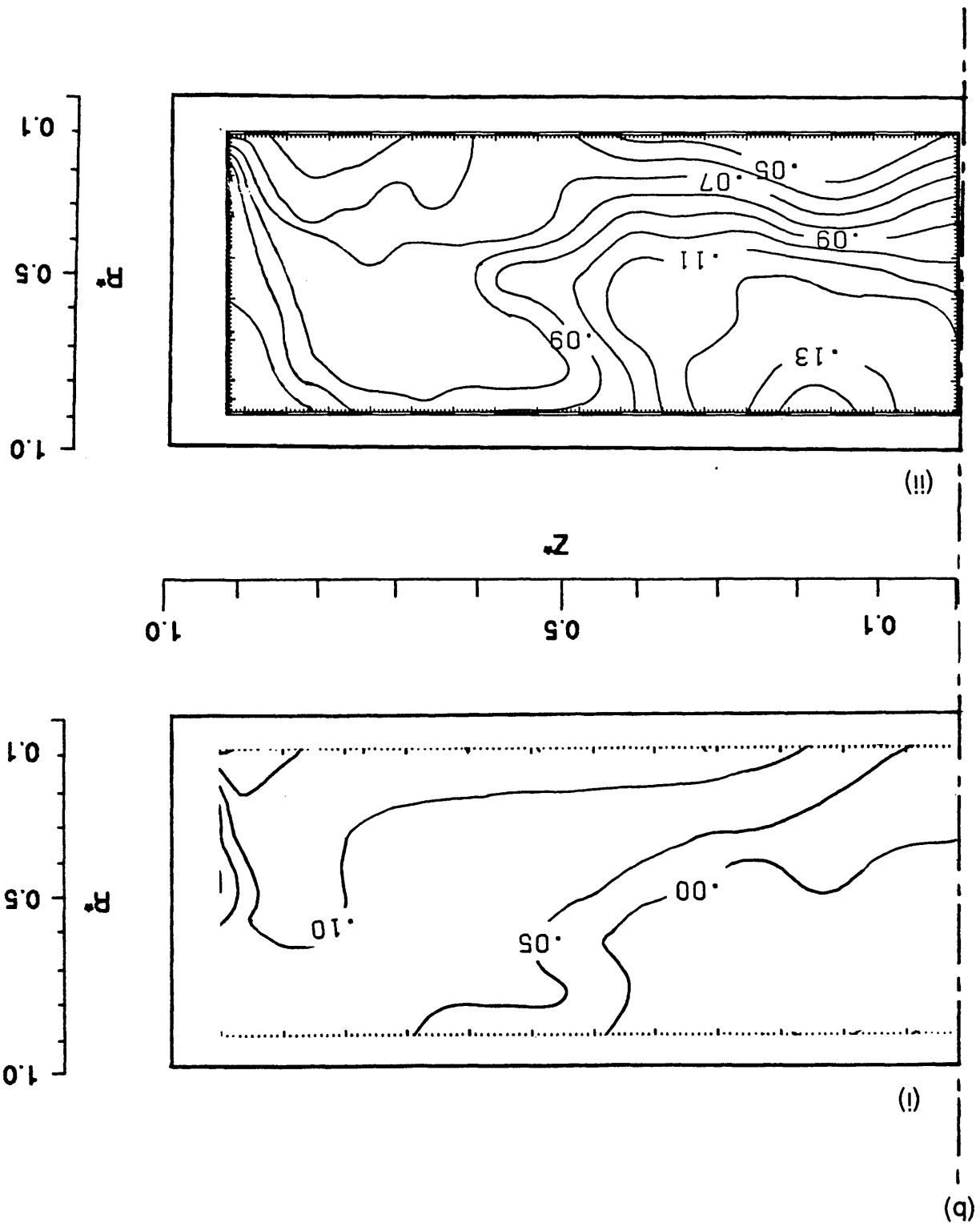
$Re_{inlet}=40,000$

(a) Streamwise Velocity

(i) Mean (ii) rms

(b) Radial Velocity

(i) Mean (ii) rms



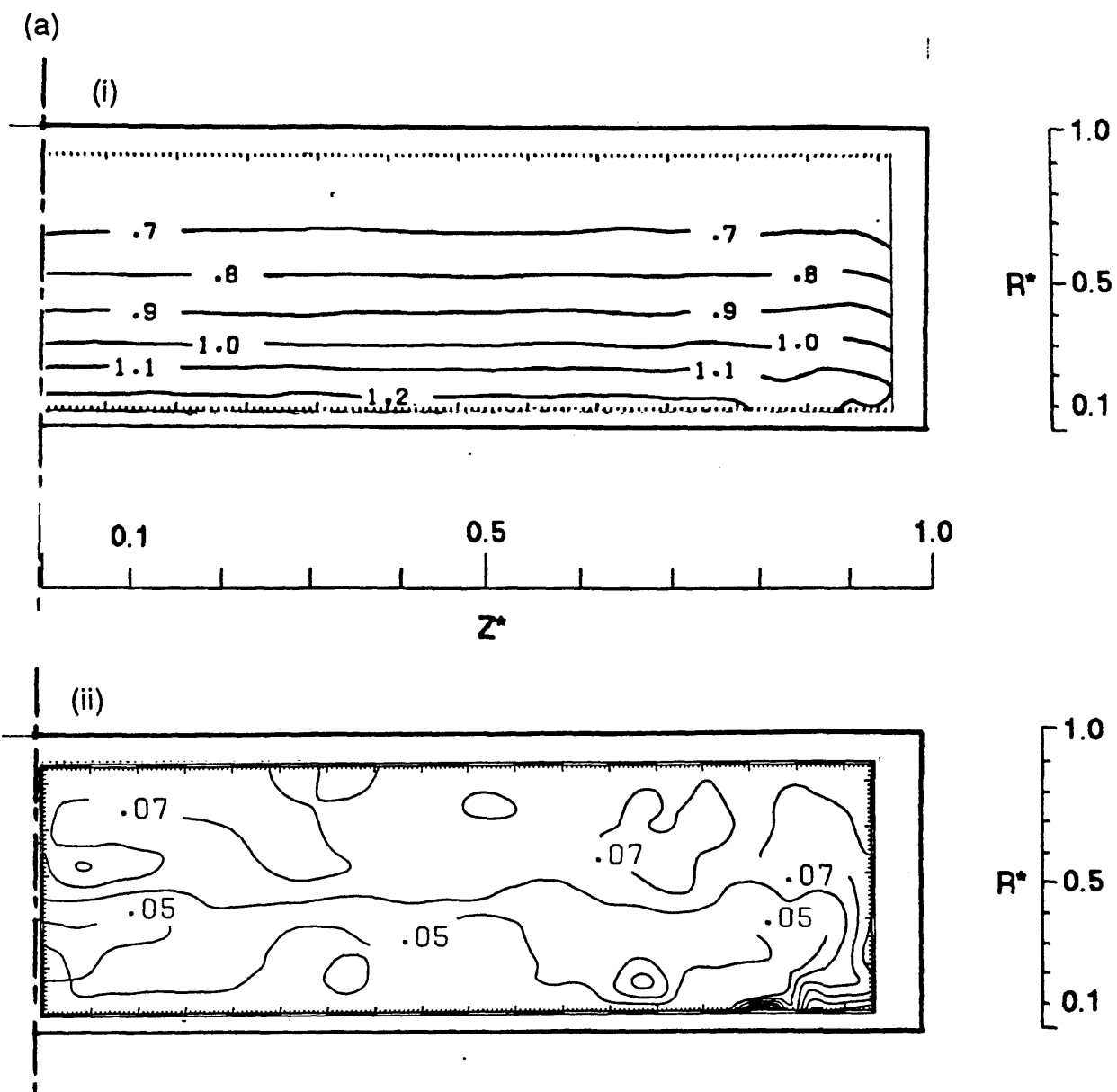


Figure 4.19 Measurements of Mean and rms Velocity at 90° Plane for

$Re_{inlet}=40,000$

(a) Streamwise Velocity

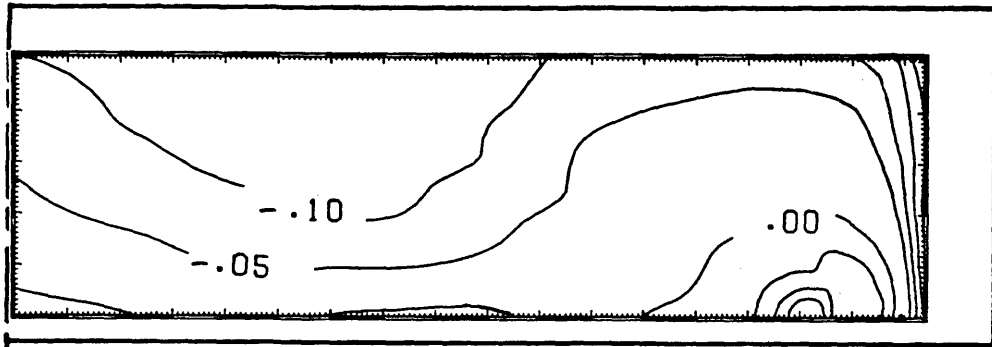
(i) Mean (ii) rms

(b) Radial Velocity

(i) Mean (ii) rms

(b)

(i)



0.1

0.5

1.0

Z^*

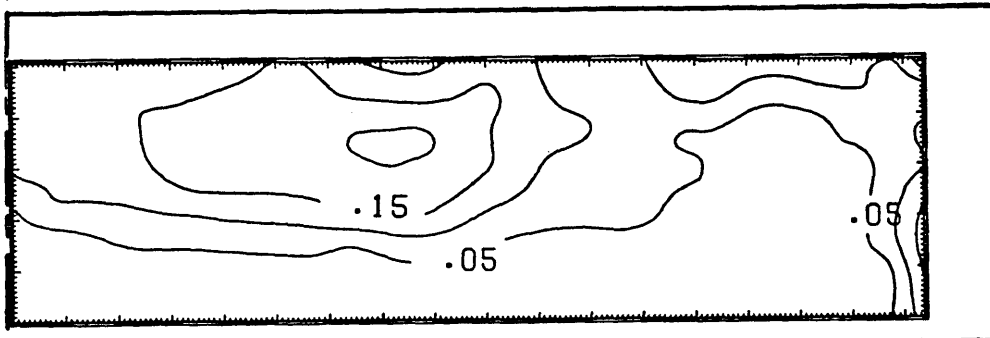
R^*

1.0

0.5

0.1

(ii)



R^*

1.0

0.5

0.1

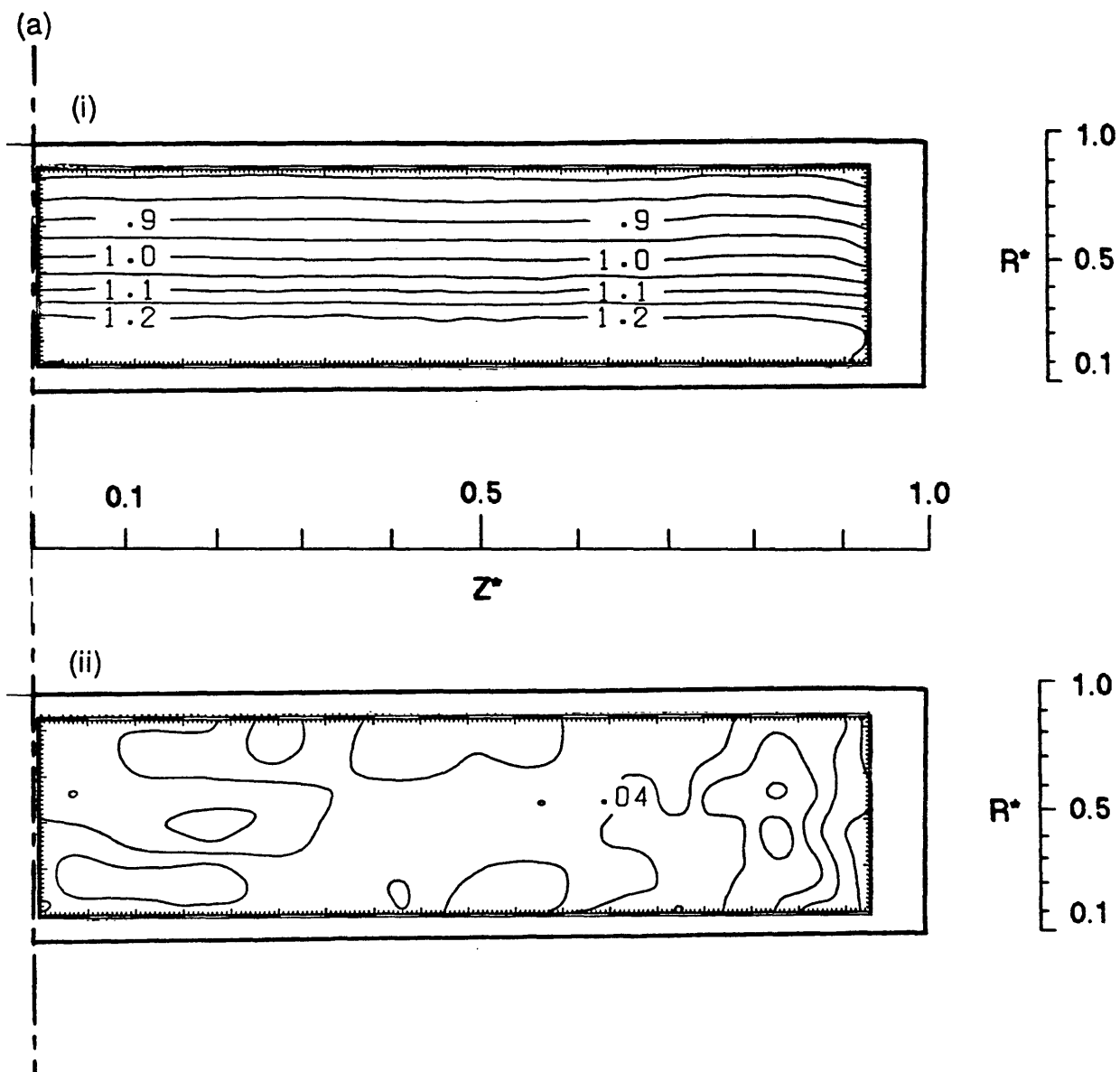


Figure 4.20 Measurements of Mean and rms Velocity at 135° Plane for

$Re_{inlet}=40,000$

(a) Streamwise Velocity

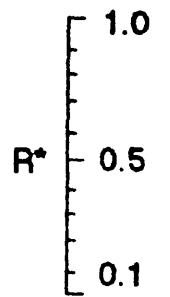
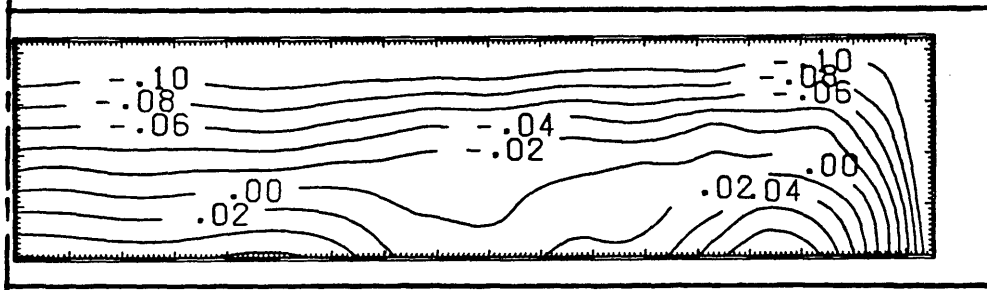
(i) Mean (ii) rms

(b) Radial Velocity

(i) Mean (ii) rms

(b)

(i)



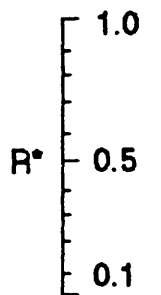
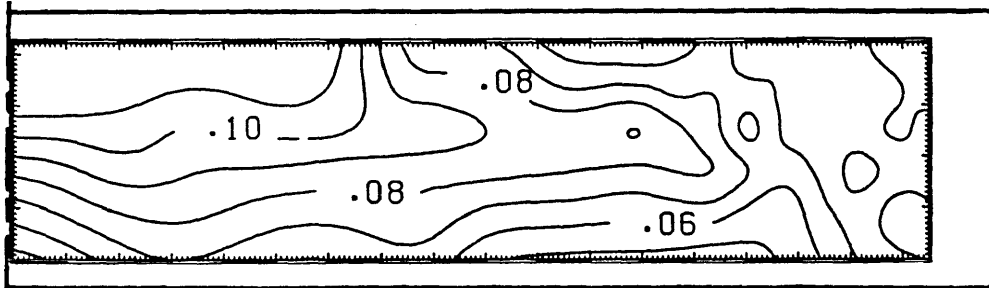
0.1

0.5

1.0

z^*

(ii)



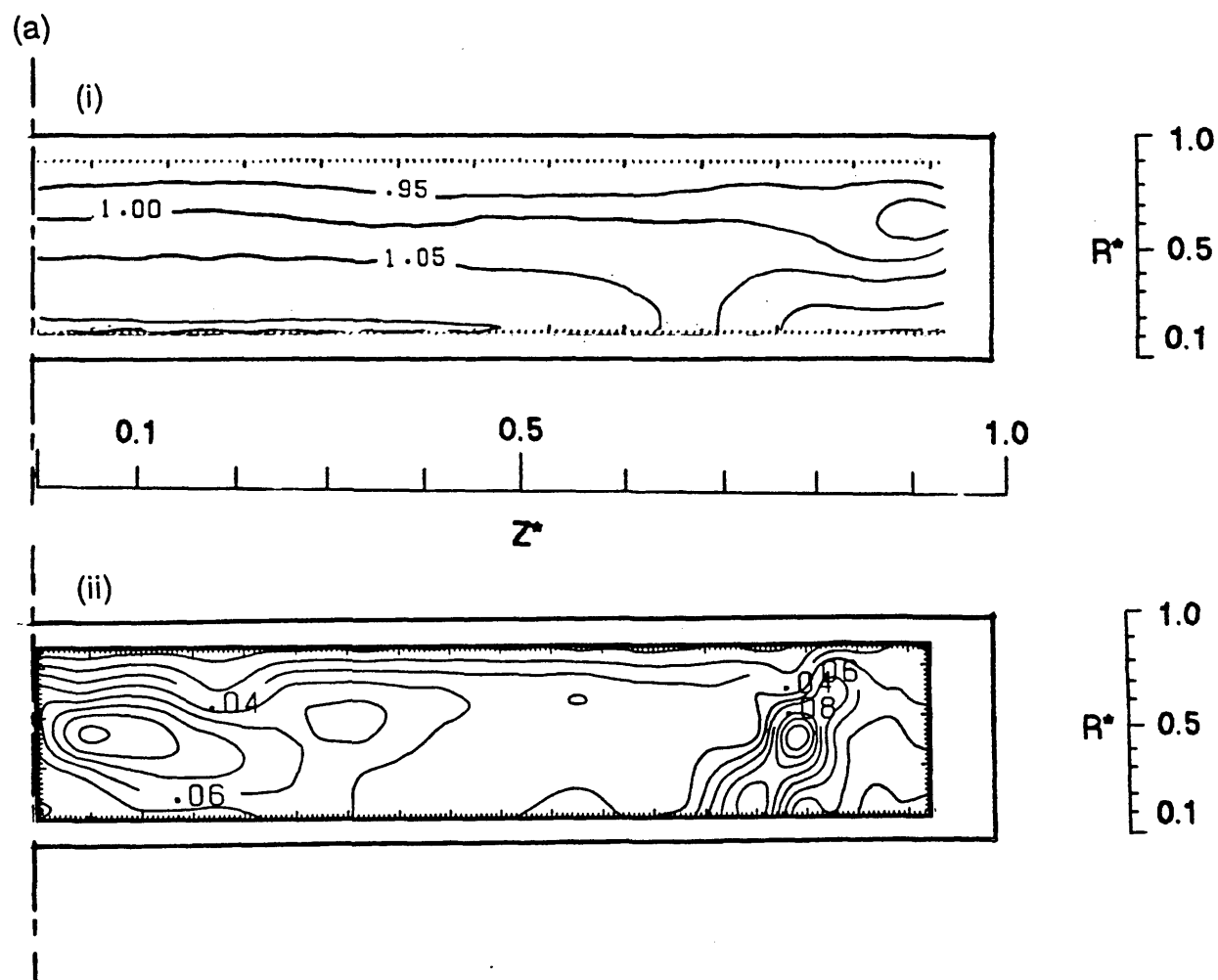


Figure 4.21 Measurements of Mean and rms Velocity at 180° Plane for

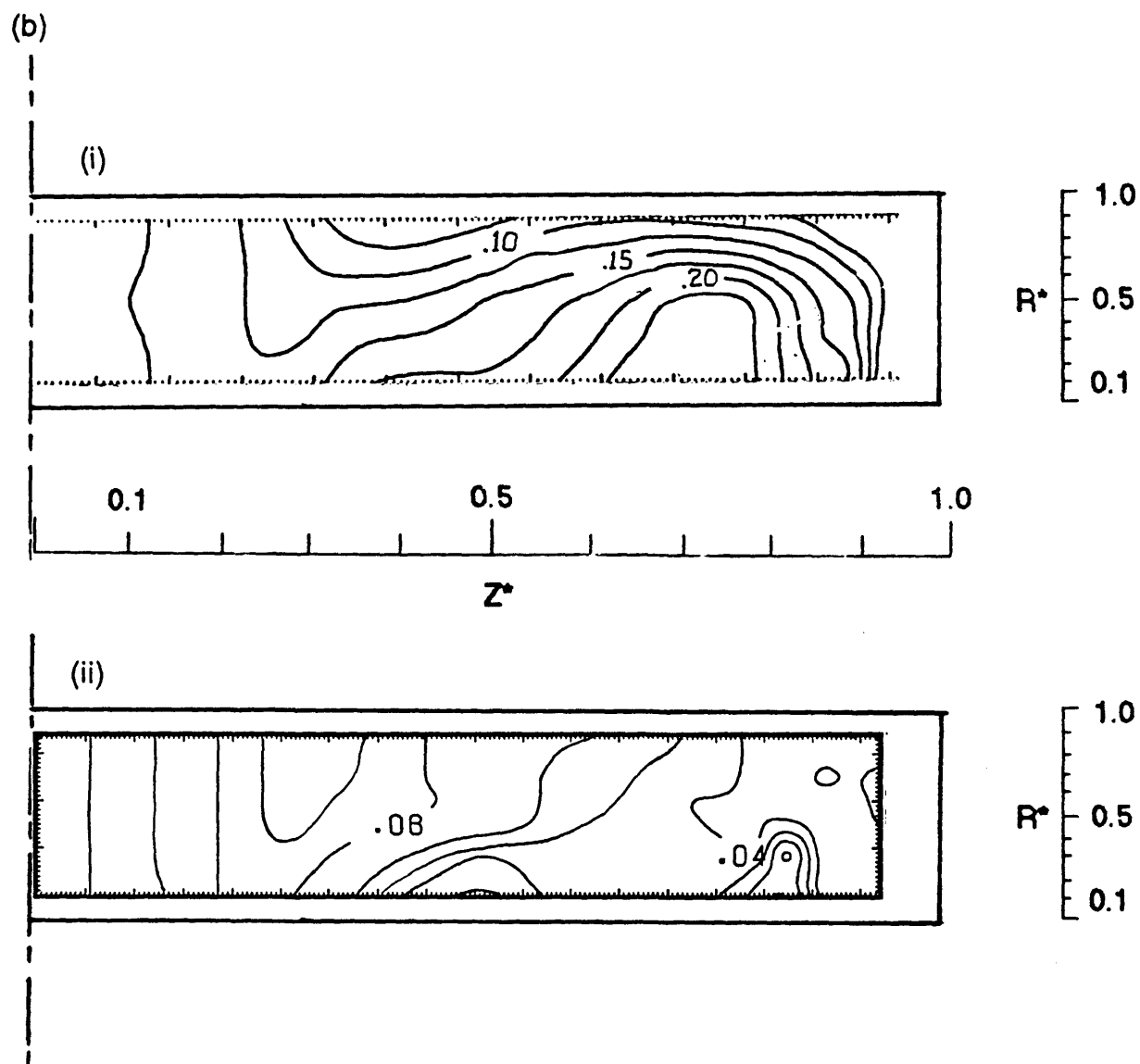
$Re_{inlet}=40,000$

(a) Streamwise Velocity

(i) Mean (ii) rms

(b) Radial Velocity

(i) Mean (ii) rms



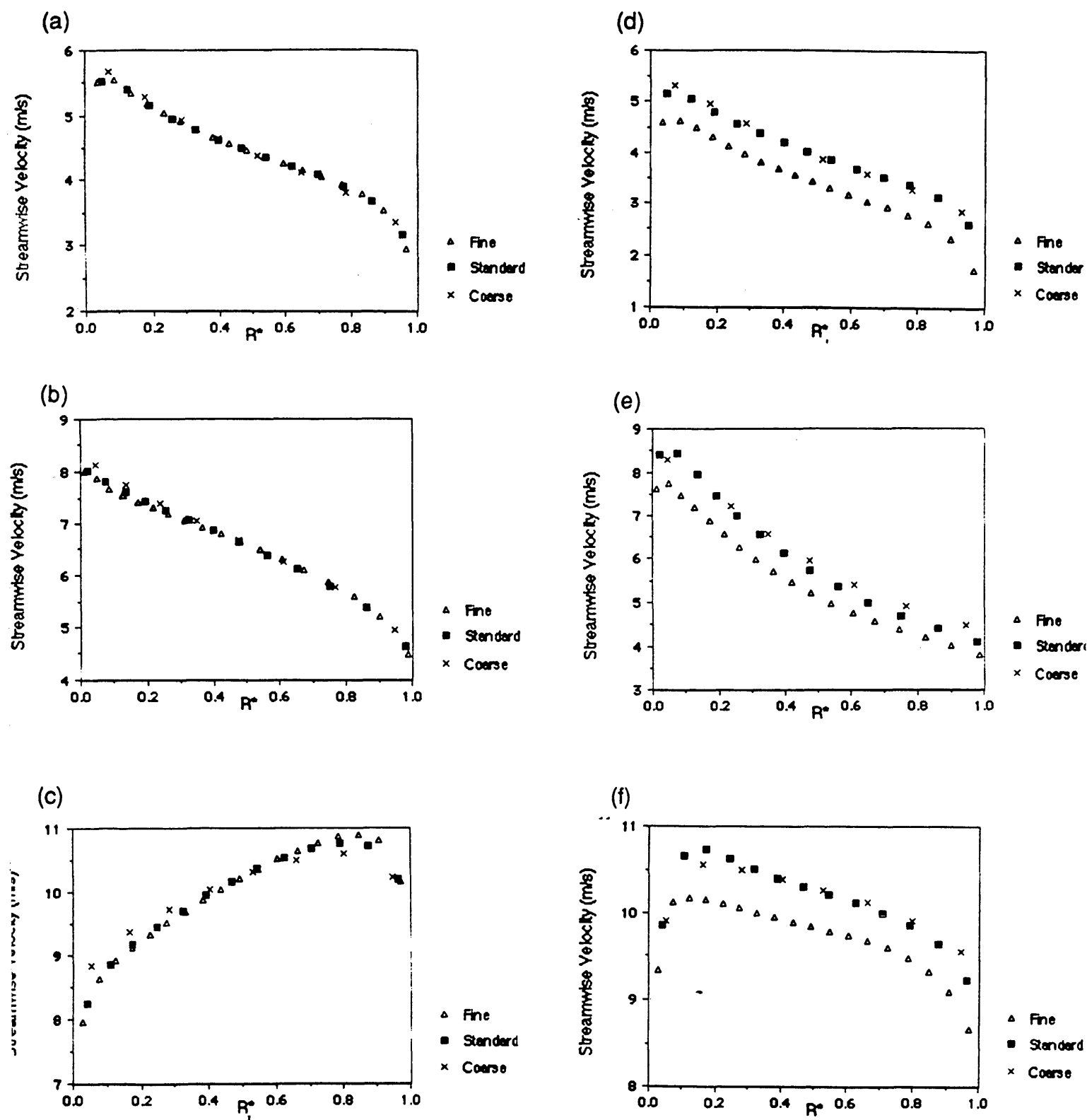


Figure 4.22 Predicted Streamwise Velocity Profile for $Re_{inlet}=15,000$
 (a) $Z^*=0.015$, $\theta=0^\circ$ (b) $Z^*=0.015$, $\theta=90^\circ$ (c) $Z^*=0.015$, $\theta=180^\circ$
 (d) $Z^*=0.985$, $\theta=0^\circ$ (e) $Z^*=0.985$, $\theta=90^\circ$ (f) $Z^*=0.985$, $\theta=180^\circ$

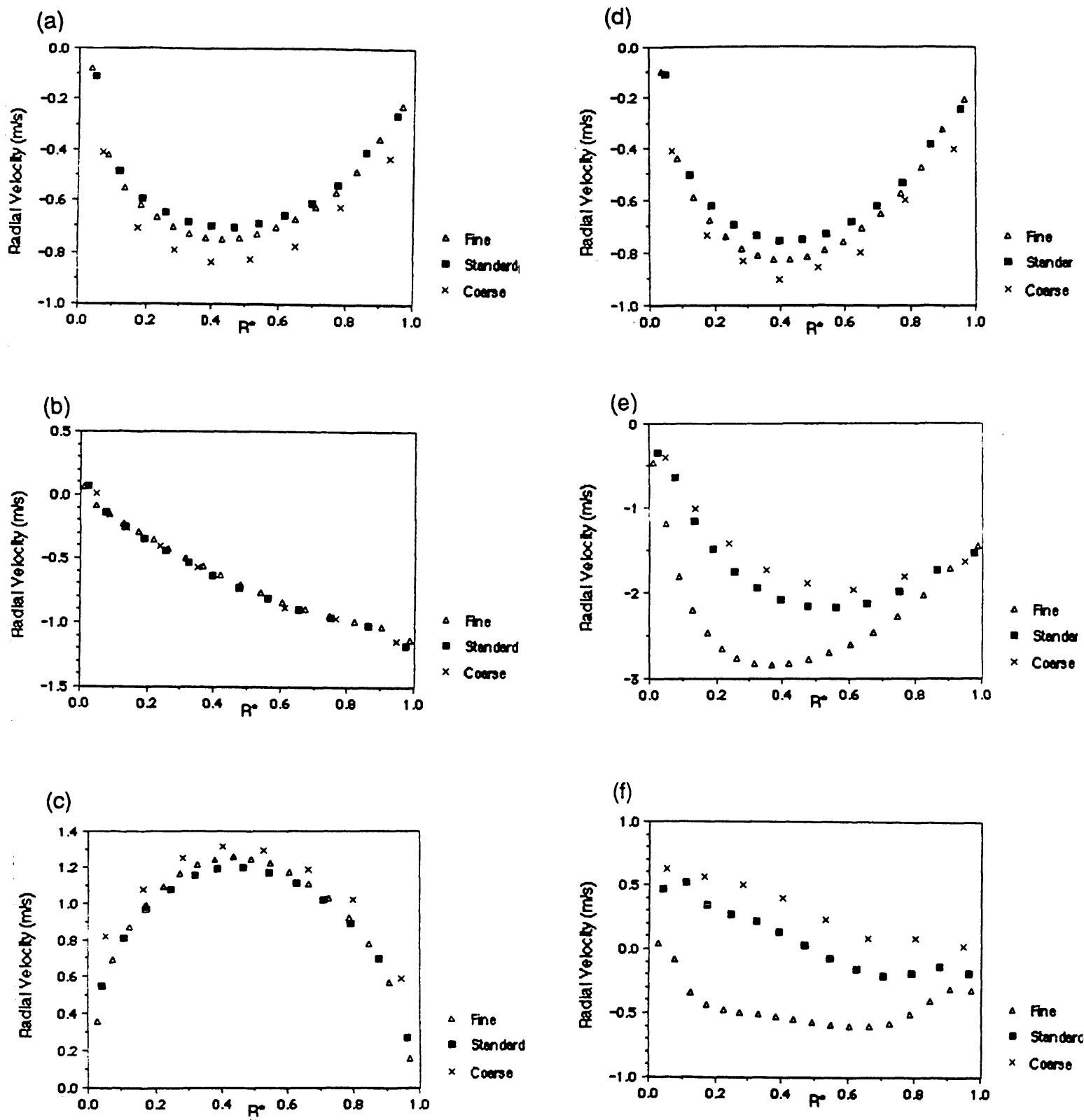


Figure 4.23 Predicted Radial Velocity Profile for $Re_{inlet}=15,000$
 (a) $Z^*=0.015, \theta=0^\circ$ (b) $Z^*=0.015, \theta=90^\circ$ (c) $Z^*=0.015, \theta=180^\circ$
 (d) $Z^*=0.985, \theta=0^\circ$ (e) $Z^*=0.985, \theta=90^\circ$ (f) $Z^*=0.985, \theta=180^\circ$

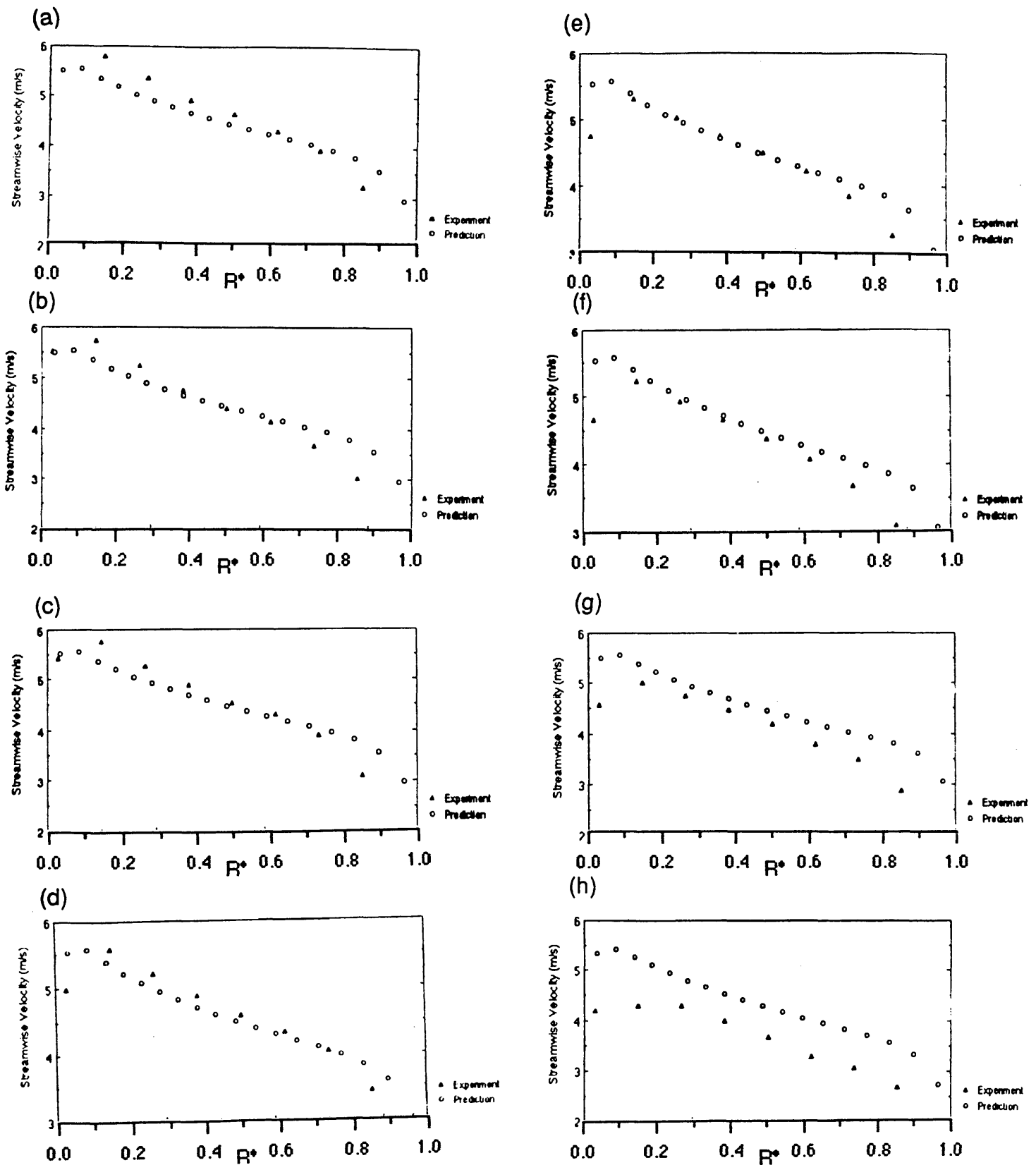


Figure 4.24 Comparison of Measured and Predicted Streamwise

Velocity at 0° Plane for $Re_{inlet}=15,000$

- | | | |
|----------------|----------------|----------------|
| (a) $Z^*=0.00$ | (b) $Z^*=0.27$ | (c) $Z^*=0.50$ |
| (d) $Z^*=0.73$ | (e) $Z^*=0.85$ | (f) $Z^*=0.88$ |
| (g) $Z^*=0.92$ | (h) $Z^*=0.96$ | |

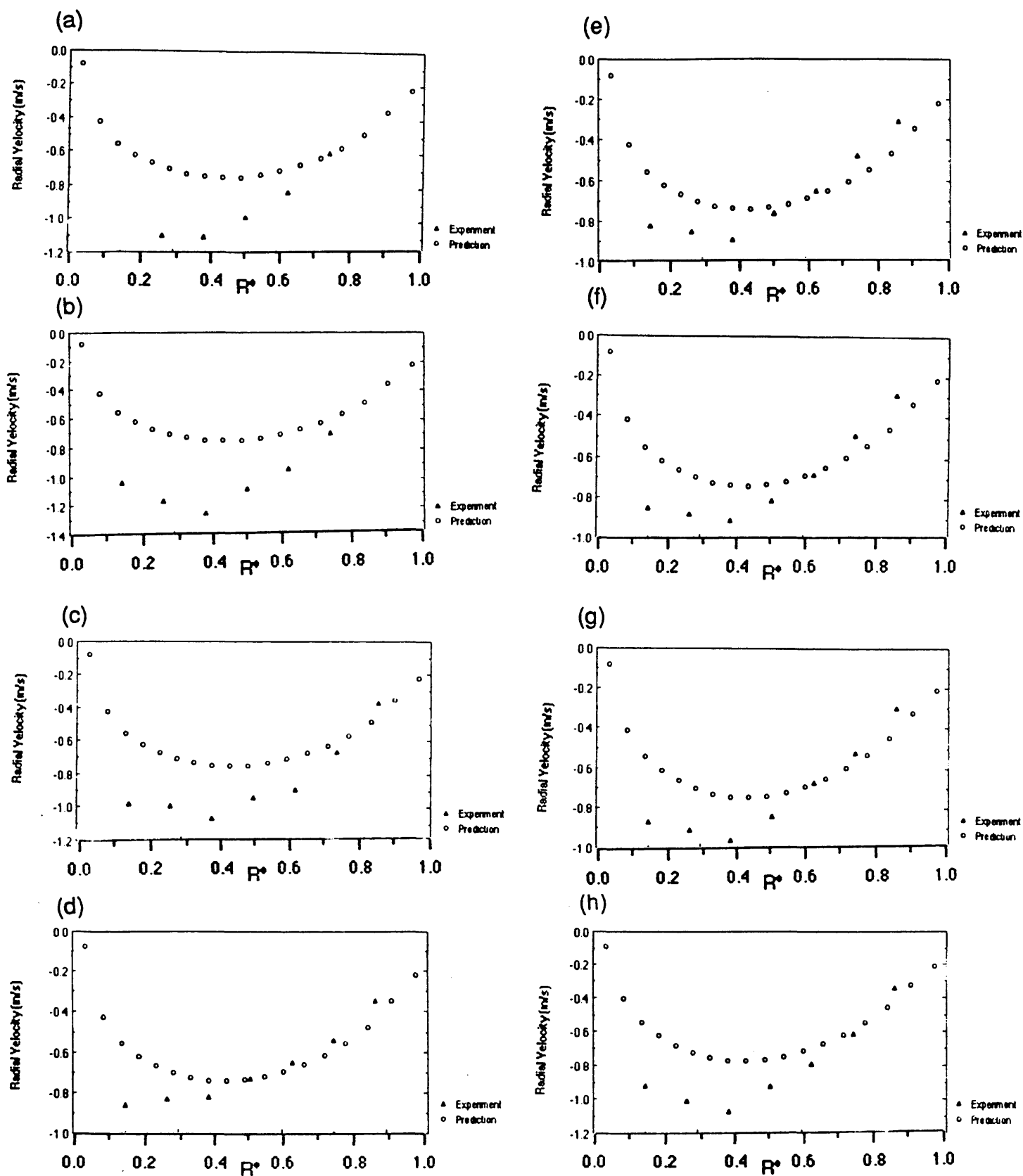


Figure 4.25 Comparison of Measured and Predicted Radial

Velocity at 0° Plane for $Re_{inlet}=15,000$

- | | | |
|----------------|----------------|----------------|
| (a) $Z^*=0.00$ | (b) $Z^*=0.27$ | (c) $Z^*=0.50$ |
| (d) $Z^*=0.73$ | (e) $Z^*=0.85$ | (f) $Z^*=0.88$ |
| (g) $Z^*=0.92$ | (h) $Z^*=0.96$ | |

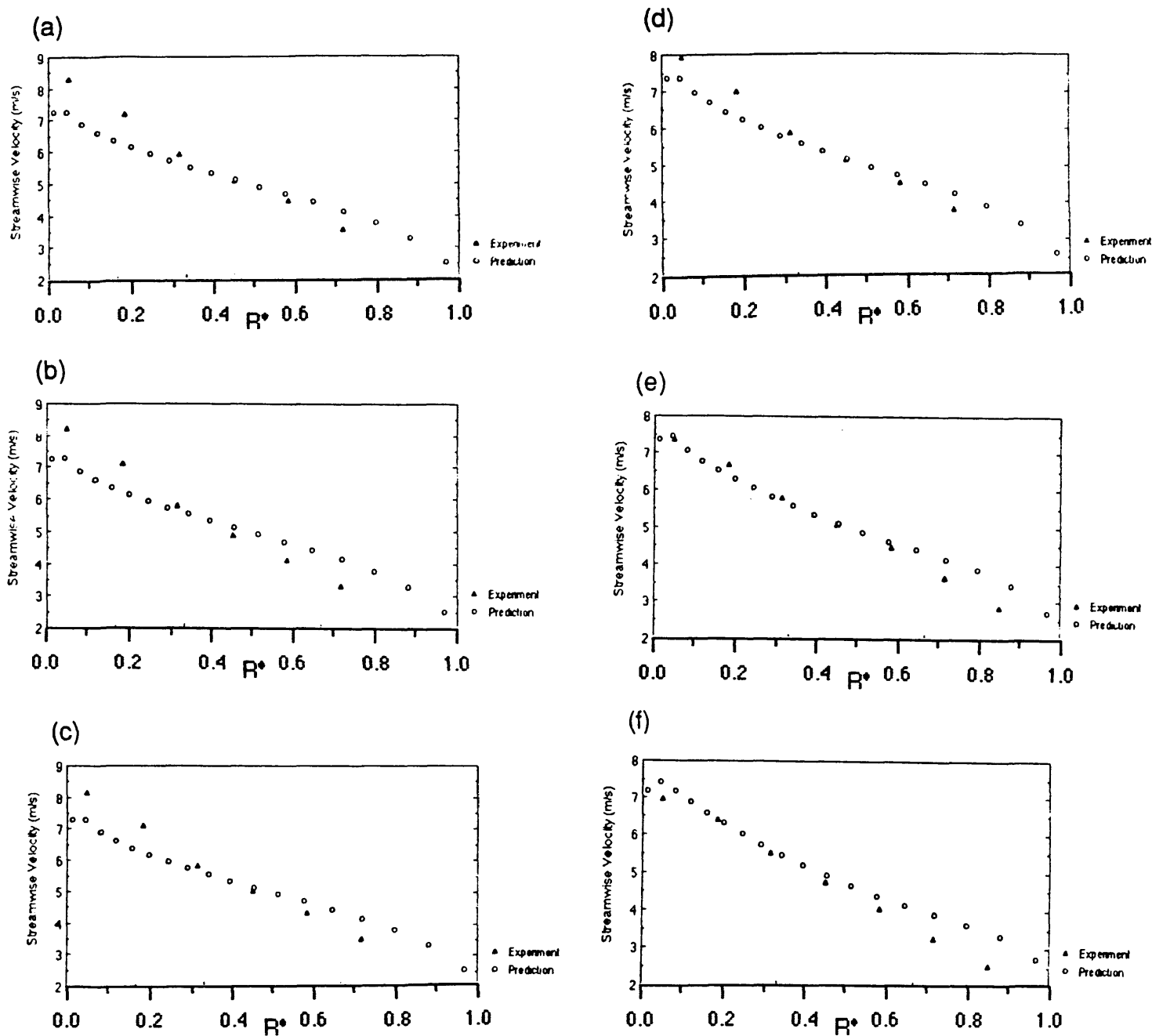


Figure 4.26 Comparison of Measured and Predicted Streamwise

Velocity at 45° Plane for $Re_{inlet} = 15,000$

- | | | |
|------------------|------------------|------------------|
| (a) $Z^+ = 0.00$ | (b) $Z^+ = 0.27$ | (c) $Z^+ = 0.50$ |
| (d) $Z^+ = 0.73$ | (e) $Z^+ = 0.85$ | (f) $Z^+ = 0.92$ |

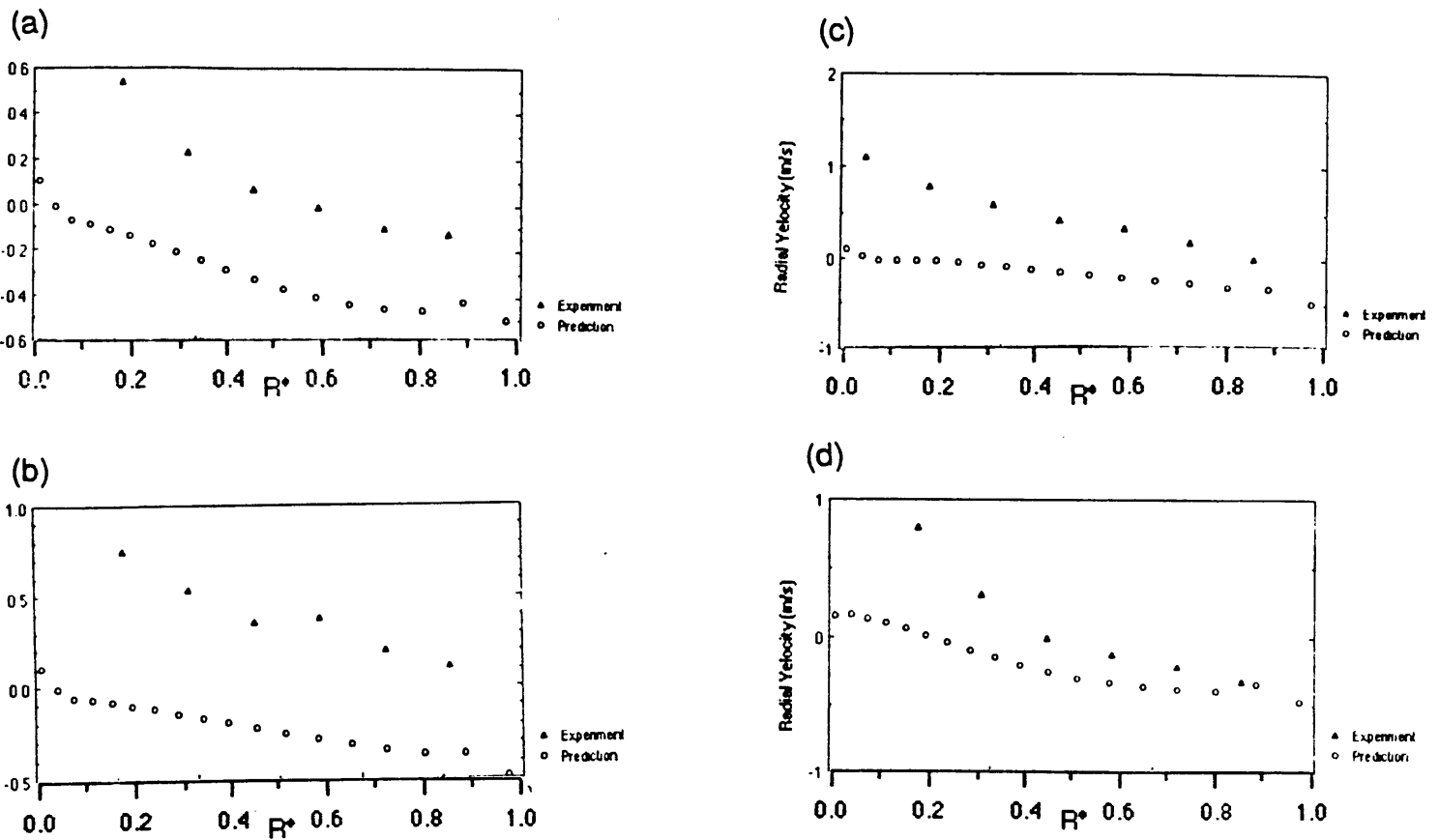


Figure 4.27 Comparison of Measured and Predicted Radial

Velocity at 45° Plane for $Re_{inlet}=15,000$

(a) $Z^*=0.50$

(b) $Z^*=0.73$

(c) $Z^*=0.85$

(d) $Z^*=0.92$

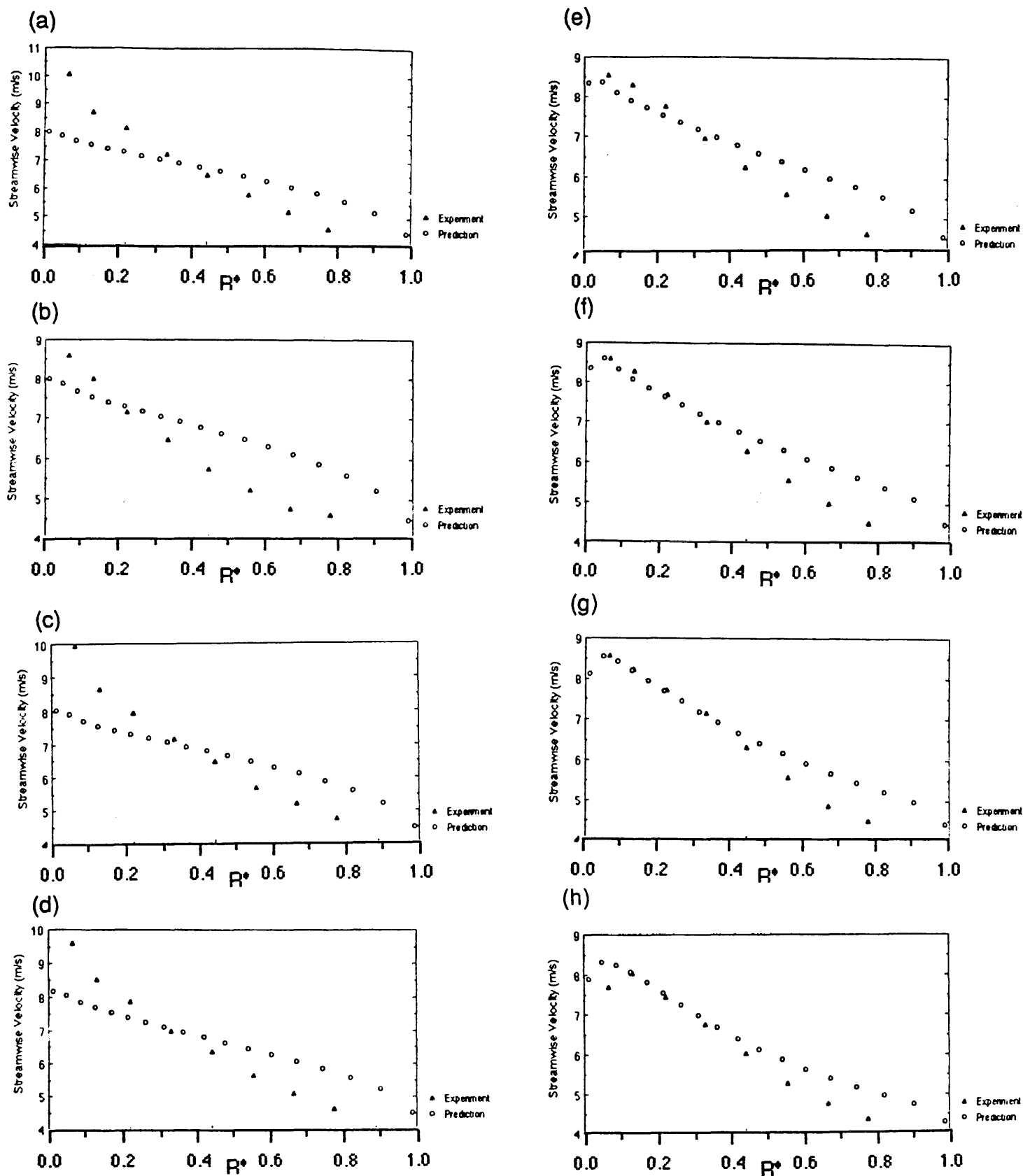


Figure 4.28 Comparison of Measured and Predicted Streamwise

Velocity at 90° Plane for $Re_{inlet}=15,000$

- | | | |
|-----------------|-----------------|----------------|
| (a) $Z^*=0$ | (b) $Z^*=0.275$ | (c) $Z^*=0.5$ |
| (d) $Z^*=0.725$ | (e) $Z^*=0.85$ | (f) $Z^*=0.88$ |
| (g) $Z^*=0.92$ | (h) $Z^*=0.96$ | |

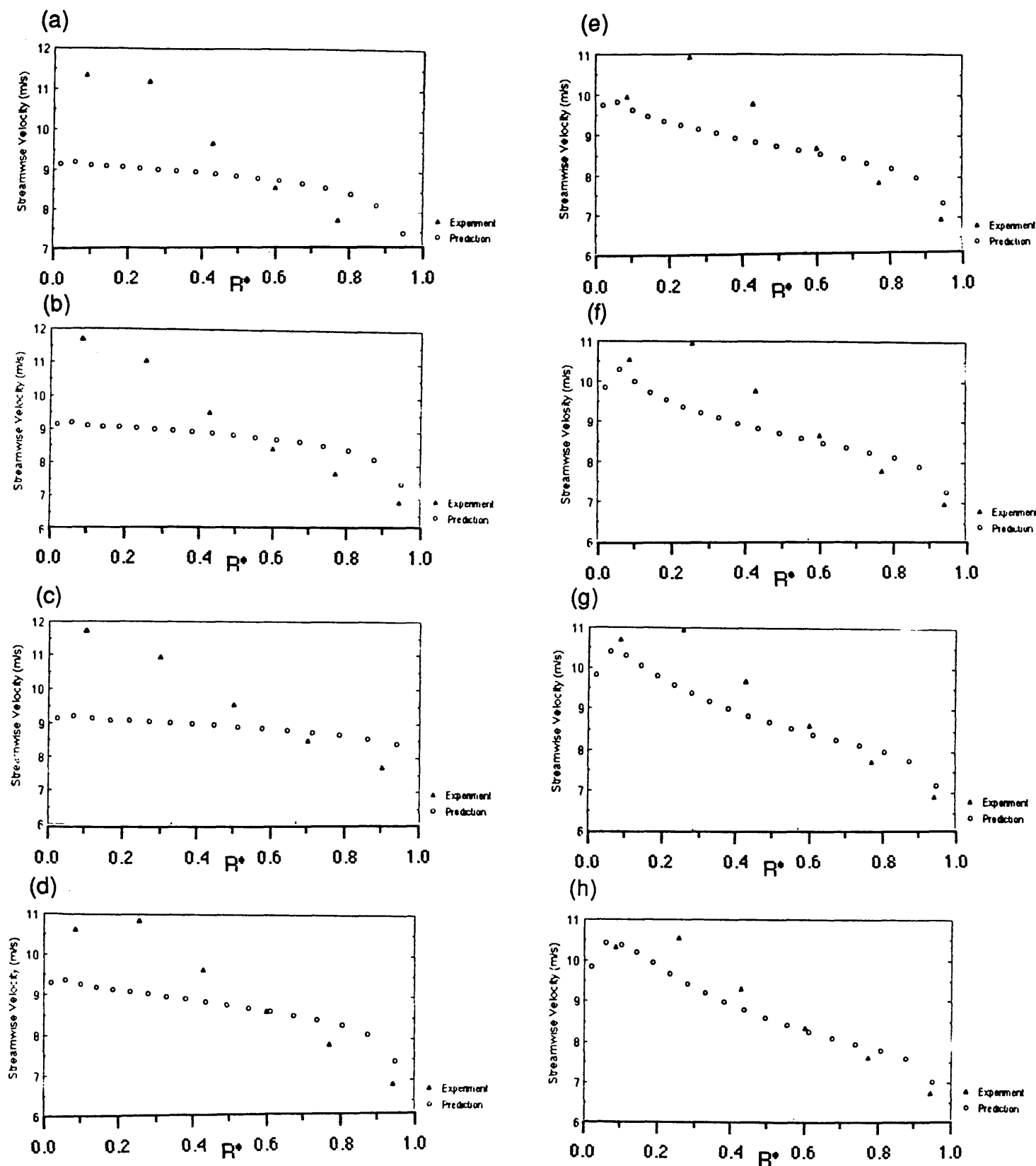


Figure 4.29 Comparison of Measured and Predicted Streamwise

Velocity at 135° Plane for $Re_{inlet}=15,000$

- | | | |
|-----------------|-----------------|----------------|
| (a) $Z^*=0$ | (b) $Z^*=0.275$ | (c) $Z^*=0.5$ |
| (d) $Z^*=0.725$ | (e) $Z^*=0.85$ | (f) $Z^*=0.88$ |
| (g) $Z^*=0.92$ | (h) $Z^*=0.96$ | |

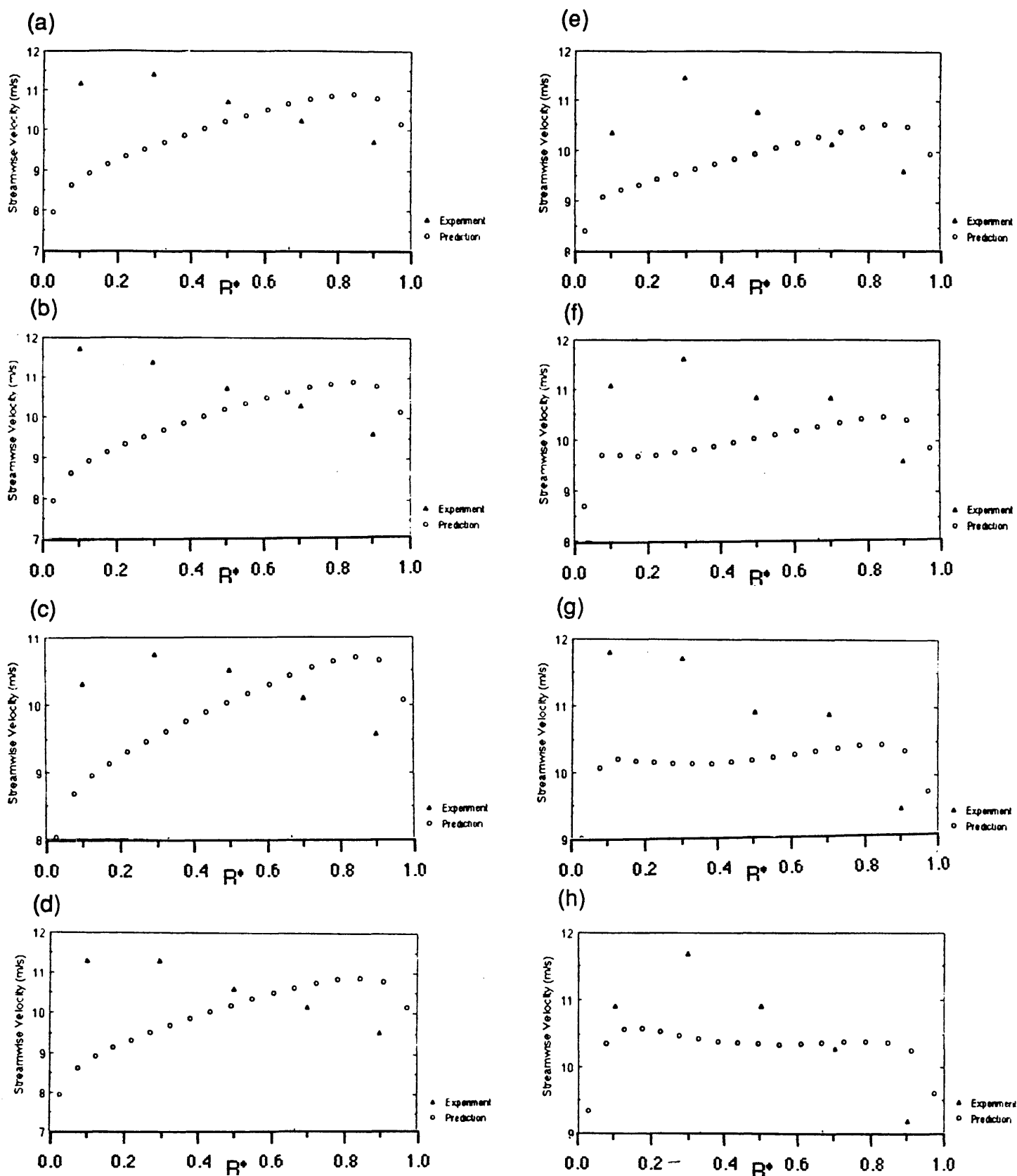


Figure 4.30 Comparison of Measured and Predicted Streamwise

Velocity at 180° Plane for $Re_{inlet}=15,000$

- | | | |
|-----------------|-----------------|----------------|
| (a) $Z^*=0$ | (b) $Z^*=0.275$ | (c) $Z^*=0.5$ |
| (d) $Z^*=0.725$ | (e) $Z^*=0.85$ | (f) $Z^*=0.88$ |
| (g) $Z^*=0.92$ | (h) $Z^*=0.96$ | |

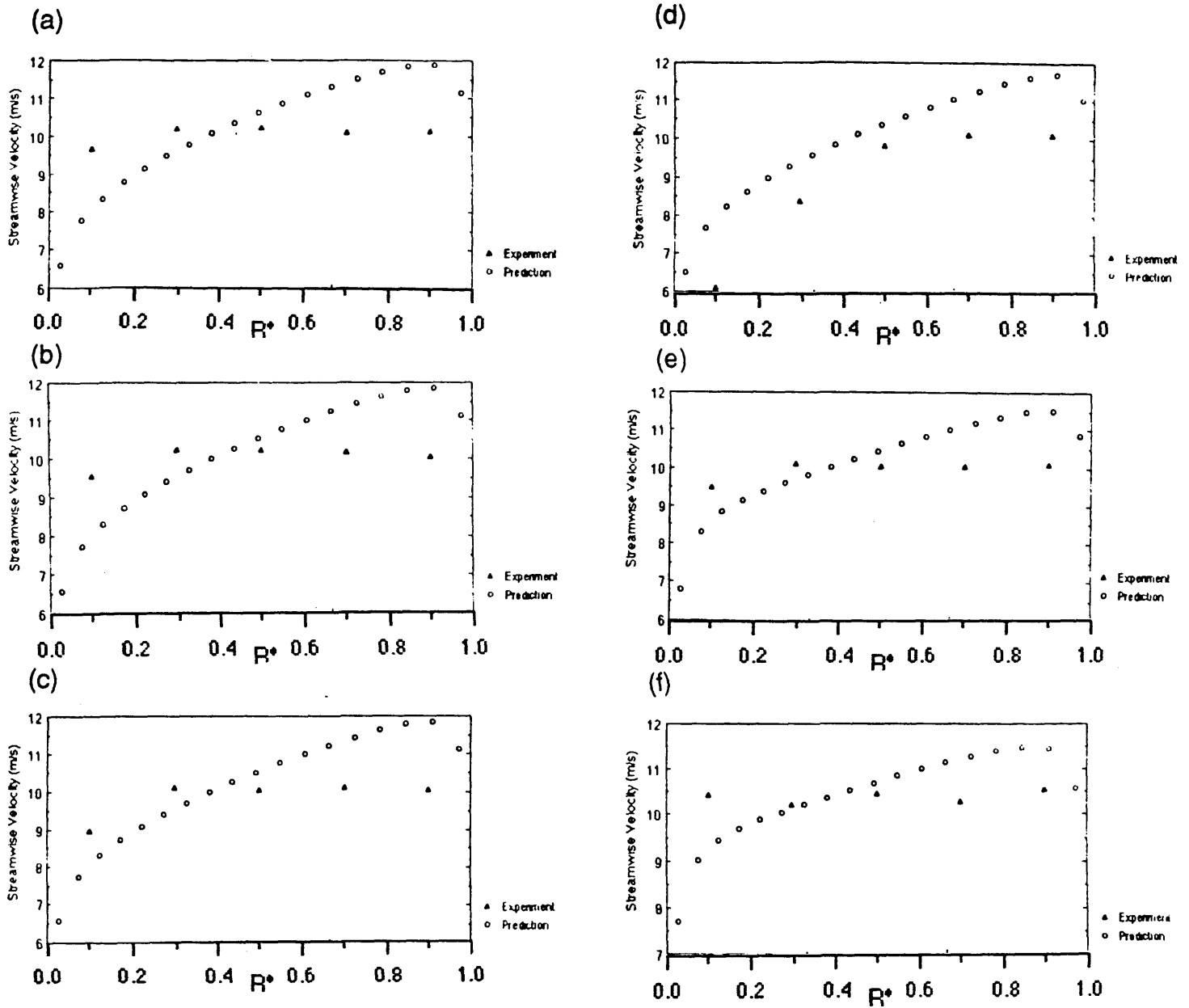


Figure 4.31 Comparison of Measured and Predicted Streamwise Velocity at Exit Plane for $Re_{inlet}=15,000$

(a) $Z^*=0.00$ (b) $Z^*=0.27$ (c) $Z^*=0.50$
(d) $Z^*=0.73$ (e) $Z^*=0.85$ (f) $Z^*=0.92$

CHAPTER 5 FLOW AND COMBUSTION CHARACTERISTICS OF A MODEL ANNULAR COMBUSTOR

5.1 Introduction

This chapter presents experimental and numerical results obtained in a model of a reverse-flow annular combustor of the type used in small gas turbines. The investigation was carried out in order to improve understanding of the flow characteristics and combustion processes which determine the performance of the combustor. The configuration of the model combustor was identical to that described by Bicen, Tse and Whitelaw (1987) and comprised two sectors, representing the primary and dilution zones, with fuel injection through a T-vaporiser, as described by Sotheran (1984). The reverse-flow nozzle has been removed and the casing replaced by a plenum chamber which allowed the same static pressure on the two surfaces of the combustor. The combustor flow consisted of a primary zone vortex driven by three film-cooling slots and limited by primary jets injected normal to the bulk flow. Additional film-cooling slots and dilution holes ensured cool walls and moderate exit plane temperatures.

Previous measurements made by Bicen, Tse and Whitelaw (1987) examined the influences of air-fuel ratio and of the alignment of the primary jets with respect to the position of the fuel injection ports and dilution holes. The variation of air-fuel ratio had little effect on the chemical properties in the primary zone since, even in the case of high air-fuel ratio, the equivalence ratio was already very high and there was insufficient air to burn more of the fuel. The temperature profiles were

found to be sensitive to the position of the primary jets which influenced the mixing processes in the primary zone to a great extent. The effect of a preheat temperature of 200K was also reported by Bicen, Senda and Whitelaw (1988) and it was shown to improve the combustion efficiency and pattern factor at the exit of the combustor. The same preheat temperature of 200K was used in the present study with kerosene ($C_{12}H_{26}$ average) as fuel, instead of natural gas (94% CH_4) as in previous studies. The results with liquid and gaseous fuels are compared to quantify the ability of the vaporiser to atomise and vaporise and, at the same time, the accuracy of temperature inferred from local enthalpy balance with liquid fuel is assessed. In addition, equilibrium temperature and composition values obtained from the measured local mean equivalence ratio quantify the extent to which the combustion processes can be characterised by mean flow properties and equilibrium reactions.

The measurement techniques and the associated uncertainties have been considered in Chapter 2 so that temperature was measured by fine-wire thermocouples with digital sampling and temperature compensation to provide the mean and fluctuating components of the gas temperature. Concentrations of UHC, CO, CO_2 , O_2 and H_2 were obtained by sampling through either a quartz microprobe or a water-cooled probe with a flame ionisation detector, infrared analysers, a paramagnetic analyser and a gas chromatograph.

Numerical simulation of flow inside a model annular combustor has been reported by Sodha (1987, 1988), based on a simplified configuration used by Toral and Whitelaw (1982), and the effect of grid refinement was studied. The results showed no improvement in the predicted velocity field with a grid refinement which doubled the number of nodes from $26 \times 15 \times 9$ to $30 \times 15 \times 15$. In continuing this numerical study of model annular

combustor, calculations were performed in the present study to provide detailed information of the flow and combustion characteristics inside a model combustor which is geometrically identical to the practical configuration. The effects of boundary conditions at air and fuel inlet ports and different flow splits through various holes and film-cooling slots are reported. In combusting flow calculations, two chemistry models, equilibrium and laminar flamelet model as described in Chapter 3, were employed to compare their merit between the two for describing the combustion processes.

The numerical procedures, described in Chapter 3, involve finite-difference approximations, with hybrid differencing of conservation equations, including those of a k - ϵ turbulence model. The turbulence/chemistry interaction was modelled by a β -function probability density function of a single conserved scalar with either the equilibrium or the laminar flamelet chemistry model. A simple castellated approximation, rather than a body-fitted mesh, was adopted to represent the combustor geometry in the present calculations since those in a simplified model combustor configuration using both castellated and body-fitted meshes, by Jones, McGuirk and Sodha (1989) and Manners (1988) respectively, showed little difference in the predicted flowfield and the former was easier to incorporate into the calculation method especially when the internal blockage of the vaporiser was introduced.

The flow configuration is described in the next section and Section 5.3 presents the details of the calculations. Section 5.4 describes the experimental and numerical results and their importance and discrepancies are discussed in Section 5.5. Finally, Section 5.6 provides the main conclusions.

The experimental results of this chapter have been published by Chow, Senda and Whitelaw (1988) and the results of the numerical simulation by Chow and McGuirk (1991).

5.2 Flow Configuration

Figure 5.1 illustrates the pipework for the supply of fuel and air to the model combustor. Air was blown at a constant rate from a centrifugal fan through an orifice plate to a 20kW preheater which increased the inlet air temperature by 200K. The bulk flow was split into two parts with orifice plates and control valves to the plenum chambers surrounding the combustor. This arrangement allowed separate control of the flow through the two sides of the combustor. The fuel was supplied through an airblast atomiser comprising two coaxial metal tubes with inner diameters of 0.5mm and 4mm as shown in Figure 5.2. The liquid fuel was pumped through the inner tube to produce a round jet of liquid which was surrounded by a coaxial and coflowing stream of high velocity air with near atmospheric pressure and a 200K preheat.

The geometry of the model combustor is shown in Figure 5.3 with details provided in Table 5.1 and is similar to that of the Gem-60 combustor which is used in small gas turbine engines. The three film-cooling slots drove a vortex in the primary zone which was limited in its extent by a row of five primary jets emerging from the lower wall of the combustor. Two additional film-cooling slots were located further downstream to provide cooling for the combustor wall. Two sets of opposed dilution holes, four holes at the top wall and two at the bottom and with an out-of-line arrangement by half pitch to the primary holes, were employed to provide a more uniform temperature profile in the exit plane.

The primary and dilution holes were plunged and guiding blocks, as described by Tse(1988), were used inside the top plenum chamber to ensure the impingement of dilution jets.

Aviation grade kerosene was used as fuel with properties listed in Table 5.2. The flow rate of preheated air through the vaporiser was the minimum required to vaporise the spray and this led to an air-fuel ratio of 2 for the mixture emerging from the vaporiser compared with 1.27 in the real combustor. The overall air-fuel ratios were set at 26 and 43 corresponding to the take-off and ground-idle conditions respectively for the Gem-60 engine with the mass flow rates scaled according to the parameter $mT^{1/2}/AP$ where m is the total air mass flow rate, T is the inlet air temperature, P is the inlet air pressure to the combustor and A is the cross-sectional area of the combustor. This parameter controls the non-dimensional pressure drop across the combustion chamber. Preliminary temperature measurements were taken in the primary zone to confirm the symmetry of flow through the T-vaporiser.

5.3 Details of Calculations

Numerical simulations were performed with the geometry of the model combustor representing the primary and dilution zones. A mesh of 40x25x65 nodes, in the x, y and z direction respectively, was necessary to resolve details of the combustor such as dilution jets and film cooling slots, Figure 5.4. It required a computer memory of 6.5 Mbyte for the calculation which is near the limit of the existing computing facilities available and further grid refinement was not possible. The number of nodes employed here was similar to that of the fine grid calculation in Chapter 4. Although a grid independent solution could not be obtained in

Chapter 4, substantial grid refinement would be necessary to improve numerical accuracy as suggested previously. Calculations by Koutmos (1985) established a minimum grid density for a flow pattern dominated by the primary and dilution jets, as in the present case, and which was efficient in terms of accuracy and cost and that used here was similar. Moreover, the calculations of Palma (1988) for isothermal flow in a typical combustor suggested that there ~~was~~ no significant difference occurred between a standard grid, with 60x30x22 nodes, and finer grids which doubled the number of nodes($\approx 75,000$ which is similar to that employed here).

The equations for velocity, pressure correction, turbulence kinetic energy and dissipation rate were solved and the boundary conditions were applied in the manner described in Section 3.3. For combusting flow calculations, two extra equations were solved for the conserved scalar and its variance and allowed the evaluation of the thermochemical state at each grid node as described in Chapter 3.

Due to the complex and partly unknown inlet boundary conditions at various film-cooling slots, dilution holes and vaporiser, the exact values used in the simulation for the input of boundary conditions were estimated, based on the inlet air distribution of the actual Gem-60 combustor as listed in Table 5.3, which allowed bulk mean velocities to be derived for all air inlet ports. The difference in the mass flow through each individual air inlet ports between the actual combustor and the model combustor was generally within 10%, as suggested by Tse (1988).

A non-reacting flow calculation was made to simulate the isothermal experiment of Tse (1988) with the vaporiser transporting air at a rate of 4.5% of the total mass flow and corresponding to a momentum flow rate similar to that given by the fuel flow at take-off condition. Passive scalar calculation was also incorporated, in non-reacting condition, where the

concentrations of the fluid from the vaporiser exit were calculated at every grid nodes. The predicted scalar value is therefore equivalent to the mixture fraction when the vaporiser fluid is the fuel where 100% represents pure fuel and 0% for pure air. A comprehensive parametric study was then carried out to study the influence of boundary conditions and the results are described in Section 5.4.2 (a). Reacting flow calculations were performed with a full equilibrium combustion model and a laminar flamelet model with results presented in Section 5.4.2 (b) and the merits of the two models are discussed in Section 5.5.

5.4 Results

This section presents results of the measurements and calculations on the Gem model combustor. The experimental results of Section 5.4.1 are divided into two parts corresponding to temperature and species concentrations measurements. Since velocity measurements, in isothermal and combusting flows with natural gas as fuel, were obtained previously by Tse (1988) in the present configuration and the flow patterns were known, they were not repeated here. Section 5.4.2 presents the isothermal results first with emphasis on the effects of boundary conditions of slots, holes and vaporiser and the reacting-flow results in the second part.

5.4.1 Measurements

Temperature measurements were obtained in the exit plane for the air-fuel ratio of 43, but not for the air-fuel ratio of 26 since the high temperature precluded the use of small diameter thermocouples. Measured and equilibrium concentration distributions, calculated from the measured equivalence ratios, are reported at the exit plane of the combustor for the

two air-fuel ratios, and in the primary zone for the air-fuel ratio of 26. The measurement grids at the exit and in the primary zone are shown in Figure 5.5. Figure 5.6 shows mean temperature measurements made in the primary zone across the whole width of the combustor with maximum temperature regions, above 1800K, in line with the two vaporiser exit. This confirms the symmetry of flow in the primary zone to within $\pm 10\%$ of the mean temperature and therefore, subsequent measurements are presented in a half cross-plane.

The measured concentrations of CO, CO₂, O₂ and H₂ were converted to a wet basis and, together with the measured concentrations of UHC, allowed the corresponding local equivalence ratio to be calculated. Since all the species containing carbon atoms (CO, CO₂ and UHC) were measured at every point in the combustor, by assuming a constant carbon-to-hydrogen ratio, corresponding to that of the fuel, and a constant oxygen-to-nitrogen ratio, corresponding to that of the air, the concentrations of those major species which were not measured (H₂O and N₂) and the local equivalence ratio could be evaluated. Mean temperature was also deduced from concentration measurements by assuming an adiabatic flow and enthalpy balance of the major species as described by Tse (1988). According to Toral and Whitelaw (1982), the equivalence ratios and enthalpy balance temperatures calculated from the species concentrations are density-weighted.

Equilibrium species concentrations and temperature were determined from the local equivalence ratio by minimising the free energy, with the assumption of chemical equilibrium and in the manner described by Gordon and McBride (1971). Due to the unrealistically high amount of CO and H₂ predicted in fuel rich region, the constrained equilibrium model of Godoy (1982) was adopted with a rich flammability limit at equivalence

ratio of 2.5. This resulted in mean species composition corresponded to chemical equilibrium for mixture of equivalence ratio less than 2.5, and for equivalence ratio greater than 2.5, the calculated equilibrium concentration would corresponding to that of chemical equilibrium at equivalence ratio at 2.5 and diluted with pure fuel.

(a) Temperature

Distributions of mean and rms fluctuation temperature in the exit plane are shown in Figure 5.7 for the air-fuel ratio of 43. Regions with maximum values exceeding 1500K, can be observed on both sides of the centre plane and correspond to the positions of the exit of the T-vaporiser. The impingement of the dilution jets pushed the two high temperature regions in the direction of negative z . The spatially averaged temperature and the pattern factor

$$\text{OTDF} = \frac{T_{\text{max}} - T_{\text{mean}}}{T_{\text{mean}} - T_{\text{inlet}}} \quad (5.1)$$

are 1320K and 0.37 respectively, based on the central 80% of the area of the exit plane. The values of rms of the temperature fluctuations are between 30 and 50K over most of the exit plane, with higher fluctuations in regions of high temperature gradient, and lead to intensity levels between 3 and 4% when normalised by the local mean temperature. This indicates that the main cause of the temperature fluctuation was the transverse temperature gradient. This phenomenon is most obvious in regions where reaction is complete as suggested by Lenz and Gunther (1980). This is further confirmed by the low values of UHC in the exit plane, Figure 5.7,

and therefore, temperature fluctuations caused by the fluctuations of reaction rate were negligible.

(b) Species Concentrations

Figure 5.8 shows the distributions of major species concentrations, on a wet basis, and calculated equivalence ratios for the air-fuel ratio of 43. Maximum values of equivalence ratio above 0.4 can be observed in the regions corresponding to the high temperature zone and with values less than 0.3 near the jet impingement. Maximum levels of UHC and CO are 0.01% (mole fraction of kerosene, $C_{12}H_{26}$ on average) and 0.3% respectively indicating that combustion had proceeded almost to an equilibrium state. This is confirmed by equilibrium concentrations, Figure 5.9, determined from values of local equivalence ratio and the assumption of chemical equilibrium. The equilibrium and measured distributions of CO_2 and O_2 concentrations are almost identical with maximum levels of 5.5% and 15% respectively while no UHC and CO can be found in the equilibrium state.

Figure 5.10 shows the distributions of species concentrations, on a wet basis, equivalence ratio and temperature evaluated from a local enthalpy balance in the primary zone with the air-fuel ratio of 26. The fuel rich region is located near the lower wall of the combustor and in line with the vaporiser exit with a maximum equivalence ratio greater than 2. High temperatures occur in regions with near-stoichiometric conditions ($0.9 < \phi < 1.2$), and occupy almost the entire width of the half cross-plane, with maximum values above 2000K. The concentrations of UHC are high, about 1.5%, in the presence of O_2 and near the lower wall, indicating that reaction has not proceeded to equilibrium in this fuel rich region ($\phi > 1.2$). Maximum values of CO and H_2 are about 5% and 2% respectively in the

lower half of the primary zone, in line with the vaporiser exit. The CO_2 and O_2 distributions are almost identical in shape to that of the enthalpy balance temperature with a maximum level of CO_2 of about 7% in the upper half of the primary zone whereas the O_2 concentration has its highest value of around 10% at the bottom edge.

Major species and temperatures determined from measurements of local equivalence ratio and the assumption of chemical equilibrium in the primary zone are shown in Figure 5.11 and comparison with the measured results shows important differences. The equilibrium temperatures, though qualitatively similar to that based on enthalpy balance, are around 200K higher for most of the region. This is due to the assumption of complete fuel breakdown in the presence of O_2 in the equilibrium calculation whereas in reality, finite amounts of UHC were present. The equilibrium values of CO and H_2 have maxima of 20% and 17% respectively and suggest that the modified equilibrium assumption is unable to predict reasonable levels of CO and H_2 in fuel rich regions ($\phi > 1.2$).

With the air-fuel ratio of 26 and in the exit plane, the measured and equilibrium concentrations are shown in Figures 5.12 and 5.13 respectively. The fuel rich zone remains in the same regions as for the air-fuel ratio of 43 and the measured maximum values of UHC and CO are 0.05% and 1.5% respectively while the maximum equilibrium concentration of CO is 0.1% in the region with maximum equivalence ratio of 0.7. The distributions for O_2 are in close agreement with the maximum value of 11% found near the jet impingement regions. Chromatography measurements of H_2 showed insignificant levels at the exit and, since the technique is time consuming, they were not made in the exit plane and this decision is supported by the equilibrium values of Figure 5.13. The lower-than-equilibrium measured values of CO_2 , with maximum of 7% compared

with 9% based on equilibrium calculation and higher-than-equilibrium measured values of CO, indicate that the CO to CO₂ reaction was limited by chemical kinetics. Equilibrium temperatures are slightly higher than those suggested by the enthalpy balance, as shown in Figure 5.14, due to the small quantities of UHC measured and this further implies that complete equilibrium has not been reached. A summary of the performance of this model combustor in terms of combustion efficiency, emission indices of CO and UHC, as described in Mellor (1976), and pattern factor is given in Table 5.4. The discussion of the effects of using different fuels, by comparing the results presented above with that of Bicen, Senda and Whitelaw (1988) and its implications are presented in Section 5.5.

5.4.2 Calculations

The numerical simulations were performed on an Amdahl 5890 mainframe computer and iteration terminated when the maximum residual, as defined in Section 3.3.1, for each of the equations became less than 10^{-4} . In isothermal simulation, this corresponded to between 600 and 800 iterations with each iteration requiring about 45 seconds of CPU time. For reacting flow calculations, about 2000 iterations were necessary to obtain a converged solution with each iteration requiring about 60 seconds of CPU time due to the two extra equations to be solved and the construction of a PDF for the conserved scalar.

(a) Isothermal Flow

The streakline plots of the flow simulation, incorporating the five cooling slots, primary and dilution holes for air supply and T-vaporiser for fuel supply, are shown in Figure 5.15 in the vertical plane at $z = -42, -28, -14, 0, 14, 28, 42\text{mm}$. At $z = \pm 28\text{mm}$, the fluid emerging from the vaporiser

exit impinges on the circular head of the combustor causing two small regions of recirculation. The extent of the primary zone vortex in the downstream direction is limited by the dilution jets as observed at $z=\pm 14$ and ± 42 mm. The recirculation downstream of the primary jets near the bottom wall at $z=0$ and ± 28 mm is caused by the interaction between the wall jets, formed as a result of the top dilution jets impinging on the lower wall, the film-cooling slot at the bottom wall near the exit and the bulk flow in the streamwise direction.

At $z=-14$ and 42 mm, the flow downstream of the regions of dilution-jets impingement is directed downwards due to the greater momentum and mass flow rates of the upper jets whereas the upward flow in the region upstream of impingement is caused by the direction of flow of the primary zone vortex. The streakline plots in the cross-stream planes at $x=45$ (primary jet plane), 74 (dilution jet plane) and 100 mm (exit) are shown in Figure 5.16. Perfect symmetry can be observed at $x=45$ mm and indicates that the influence of the asymmetrically positioned dilution jets did not extend upstream to the primary zone. The flow pattern in the dilution zone is dominated by the dilution jets and their influence extent to the exit of the combustor where strong downward velocities, above 10 m/s, can still be found at planes in line with the four top dilution jets. Strong cross-stream velocities, with a maximum value of the same magnitude as the bulk mean streamwise velocity of 6.7 m/s, prevails downstream of the dilution jets and indicates good mixing of cool air from the dilution jets and the combustion products from the primary zone in reacting condition.

The results of the predicted passive scalar distributions at $z=0$ and 28 mm, Figure 5.17, shows high concentrations of vapouriser fluid in the bottom half of the combustor upstream of the row of primary holes and a maximum mixture fraction of 70% in the centre plane of the combustor,

$z=0\text{mm}$. At the exit of the combustor, the distribution is close to uniform, with a maximum value of 7%, apart from regions close to the top and bottom wall where the value of mixture fraction drops to zero because of the film-cooling slots.

Measurements by Tse (1988) in an identical model combustor revealed that the presence of combustion does not alter the general nature of the flow and the isothermal results can be used to indicate the trends caused by variations in inlet condition. Therefore, a parametric study, described in the following subsections, in non-reacting flow, which required much less computing resources than reacting flow calculation, would be essential to assess the influence of inlet boundary condition and, at the same time, quantify the uncertainties caused by prescribing unknown boundary conditions.

Effects of Boundary Conditions of Dilution Jets

When the streakline plots of Figure 5.15 were compared with the flow visualization of Tse (1988), differences were observed, particularly around the region of dilution-jet impingement and it was concluded that, since the top wall of the combustor near the dilution holes was inclined at an angle of 15° to the horizontal plane, the direction of entry of the top dilution jets was likely to be at a small but finite angle to the vertical. Calculations were then performed with top dilution jet angles to the vertical of 5° , 7.5° , 10° , 12.5° and 15° to study the influence of the angle on the flow pattern and Figure 5.18 shows the streakline plots at $z=-14\text{mm}$ for the five cases, which can be compared with Figure 5.15c in which the top dilution jets enter vertically. There is very little upstream influence of the angle of the dilution jet to the primary zone. With a jet entry angle of 5° , the two jets impinge without slip and, as the angle increases, slip occurs

and the location of maximum velocity at the exit moves towards the top wall. When the angle exceeds 10° , no impingement is observed.

Figure 5.19 illustrates the effects of the angle of the top dilution jet on impingement on the bottom wall, at $z=-42\text{mm}$. With the angle of 15° , the vortex at the bottom wall is pushed upstream beyond the film-cooling slot at the bottom wall but the maximum velocity at the exit remains at about 10m/s near the bottom wall. The characteristics of the small recirculation at $z=-28\text{mm}$, caused by the wall jets and the film-cooling slot at the bottom wall, are also altered as shown in Figure 5.20. For jet entry angles greater than 10° , the location of jet impingement moves upstream beyond the film-cooling slot and therefore no recirculation is observed beyond the slot. This also results in a more uniform exit velocity profile at this plane. The predictions with jet entry angle of 7.5° leads to results closest to those of the flow visualization, as shown in Figure 5.21, and this particular boundary condition for the entry angle of the top dilution jets was employed in subsequent calculations.

The effect of jet entry angle is important to jet impingement in the dilution zone but, with top dilution jets impinging on the bottom wall, at $z=-42$ and 14mm , no difference is observed with angle less than 10° , as shown in Figure 5.19. Therefore, the investigation was not extended to the primary jets since a jet entry angle exceed 10° would be unrealistic in both the model combustor and the actual flame-tube.

Effects of Boundary Conditions of Film-Cooling Slots

Calculations were carried out with different flow splits through air inlet of the primary zone to investigate their effect on the flowfield. In the first case, the mass flux through the backward facing film-cooling slot at the top wall, slot 3, was increased by 50% with the extra mass flow taken

from the primary holes to maintain a constant overall mass flow rate. The streakline plots at $z=0$ and 28mm, and $x=45, 74$ and 100mm, Figure 5.22, show that the primary jets are deflected further downstream and impinge at the top wall at a distance of about 1 jet diameter further downstream, as compared with previous calculation, due to the lower momentum of the jets. The centre of the primary zone vortex remains at the same location and no observable difference in the velocity field in the dilution zone is predicted, by comparing Figures 5.22d and 5.22e with Figures 5.16c and 5.16d. Figure 5.22c shows that the wall jets, caused by the impingement of the primary jets to the top wall, do not coalesce to form a downwash fountain as in Figure 5.16b.

Figure 5.23 shows the streakline plots for which the mass flux through the film-cooling slot at the circular head, slot 4, was increased by 50% instead of slot 3 as in the first case. There has been no change in the velocity field as compared with the first case.

A third case was tested with the mass flux through slots 3 and 4 each increased by 35% and the flow through primary jets reduced accordingly. Figure 5.24 shows the results in terms of streakline plots and no primary jet impingement is observed with the jets being deflected further downstream. However, in the dilution zone for $x>60$ mm, the flow pattern remains the same as in the previous two cases. The calculation for the third case indicates that most of the air from the primary jets flowed directly to the dilution zone whereas in the previous cases, primary jet impingement at the top wall was predicted and about 50% of the air from the primary jets, calculated by integrated the streamwise velocities, flowed upstream and entered the primary zone vortex. Therefore, a further reduction in mass flow through primary jets would have a more significant effect because, first, it represents the case where air from primary jets

would not flow upstream and then enter the primary zone recirculation and, secondly, the effect on the ability of the primary jets to limit the extent of the primary zone recirculation would be insignificant. However, this flow split would be unrealistic for combustor operation, since it means that air flow through the primary jets is less than that through any single film-cooling slot and, therefore, no further attempt was made to quantify the effect. It can be concluded, from the results presented above, that the primary zone characteristics, which determine the mixing of fuel and air, are insensitive to a limited range of variation of flow split to the primary zone as long as the mass flow in the primary zone vortex remained unchanged.

Effects of Boundary Conditions of the Vaporiser

The inlet boundary conditions of fluid emerging from the vaporiser are essential in determining the distribution of fuel in the combustor. The high predicted values of mixture fraction in the centre of the combustor, at $z=0\text{mm}$, were due, in part, to the assumption that the vaporiser flow took the form of straight and uniform jets. The two streams of vaporiser fluid from the vaporiser exit at $z=\pm 28\text{mm}$, on hitting the circular head, would spread as two wall jets, with equal strength towards and away from the centre plane and impinge at $z=0\text{mm}$ causing a high concentration of fuel.

A non-uniform velocity profile was introduced, based on probe measurements made at the exit of the vaporiser by Tse (1988). The velocity profile had a minimum of less than 20m/s , at a position 1mm from the inside edge of the vaporiser exit and increased to a maximum value of above 50m/s at 1mm from the outer edge. The results are shown in Figure 5.25 for the passive scalar distribution at $z=0$ and 28mm . Since more

vaporiser fluid was transported away from the centre, a maximum mixture fraction of less than 30% resulted in the centre plane.

A second component of velocity, which directed vaporiser fluid away from the centre plane, was introduced to study its effects on the mixing processes. The values employed were 15 and 25% of the bulk mean velocity at the vaporiser exit and the results are shown in Figures 5.26 and 5.27 respectively. The maximum value of mixture fraction at $z=0\text{mm}$ is reduced to below 20 and 15% although the distributions are the same at $z=28\text{mm}$. The change to the vaporiser flow did not, however, affect the distribution of passive scalar downstream of the row of primary jets. This study reveals the sensitivity of primary zone fuel distribution to the inlet boundary conditions of the fuel streams and interpretation of further computational results has to take into account of the boundary conditions prescribed at fuel and air inlet ports; alternatively, the numerical simulation has to extend to predict the flow in the external annulus as well as the internal flowfield.

(b) Combusting Flow

Calculations were performed with an equilibrium and laminar flamelet combustion model with methane as fuel and no preheat, which was the operating condition of the experiment performed by Tse (1988) where the problem of liquid fuel combustion and vaporisation can be avoided. The air-fuel ratio was set at 29 which corresponds to the take-off condition of the Gem combustor. The variations of major species concentrations given by the two models with respect to mixture fraction are shown in Figure 5.28. The equilibrium model gives much higher concentrations of CO, with a maximum of 15%, and of H_2 , with a maximum of 25%, in fuel rich regions, compared with those given by the flamelet values of 6% and

5% for CO and H₂ respectively. The amount of UHC is higher with the flamelet model but no O₂ is predicted for mixture fractions greater than 0.1 in both models. The effect of adopting different chemistry models to the velocity field is only accomplished through the different density values, given by the two models, in the conservation equations. Figure 5.29 shows that the percentage difference in the instantaneous density field for the two models is less than 10% with a maximum greater than 25% when the mixture fraction less than 0.05. Moreover, the effect of fluctuations is to reduce the difference in mean gas density given by different models, as suggested by McGuirk (1987), therefore, the difference in the predicted velocity and mean scalar field, but not the species concentrations, would generally be less than 10%.

Calculation of Combusting Flow with Equilibrium Model

Figure 5.30 shows streakline plots at $z=-42, -28, -14, 0, 14, 28,$ and 42mm which can be compared with the isothermal flow of Figure 5.15. Combustion increases the forward and backward velocity maxima in the primary recirculation zone from 12 to 30m/s. The position of the centre of the primary zone vortex, which was at the mid-height of the combustor in isothermal flow at $z=\pm 14$ and $\pm 42\text{mm}$, has moved towards the upper wall by about 10mm which is a quarter of the combustor height, in combusting flow. The nature of the dilution jet impingement has also been altered and no slip is evident. The maximum exit velocity has increased from 15 to above 40m/s for combusting flow because of the lower gas density due to reaction. Figure 5.31 shows the velocity field for cross-stream planes at $x=45, 74$ and 100mm and that the diffusion of incoming air jets has increased due to the temperature difference between the bulk flow and the relatively cool air jets. The magnitude of the cross-stream

velocity at the exit has increased, as compared to isothermal flows, with a maximum value of 20m/s which is about 70% of the bulk mean exit velocity.

The distributions of concentrations of major species are shown in Figure 5.32 for $z=-28$, -14 and 0mm , and in Figure 5.33 for $x=36\text{mm}$. High concentrations of CO and H_2 , intermediate species of combustion, are predicted in the primary zone with maximum equivalence ratio of above 4.0 at $x=36\text{mm}$, at the lower half of the combustor in line with the vaporiser exit with maximum values of 9 and 16% respectively. This is due to the assumption of complete breakdown of fuel with equivalence ratios of less than 2.5 and there is insufficient oxygen for complete combustion. Downstream of the primary jets for $z=-28$ and 0mm , the CO and H_2 are then converted to CO_2 and H_2O with the extra air supplied and at $z=-14\text{mm}$, the region of high concentrations of CO and H_2 has extended to the plane of the dilution jets. At the exit, the concentrations of CO and H_2 are negligible due to the assumption of chemical equilibrium and relatively low values of equivalence ratio.

Calculation of Combusting Flow with Laminar Flamelet Model

Figures 5.34 and 5.35 shows the distributions of major species concentrations for $z=-28$, -14 , 0mm and $x=36\text{mm}$ respectively when a laminar flamelet combustion model replaced the equilibrium model. The distributions of equivalence ratios are similar in the two cases. The fuel rich region in the primary zone, given by the flamelet model, occupies a larger area but with a slightly lower maximum value of equivalence ratio of 3.5 as shown by comparing Figure 5.33 and 5.25 due mainly to the lower densities given by the flamelet model in regions of high equivalence ratios. Hence, for the same mass flow rate, a higher momentum flux was

predicted near the exit of the vaporiser and the fuel-air mixture would then be dispersed to a greater extent on hitting the combustor head. Maximum levels of CO and H₂ in the primary zone, are 4.0 and 3.5% respectively whereas at the exit, average values of less than 0.3% are predicted. The resulting velocity field is close to that given by equilibrium model as shown in Figure 5.36 for $z=0$ and 28mm and $x=45, 76$ and 100mm which can be compared with Figures 5.30d and f and 5.31. The major differences between the two models are shown in the CO and H₂ mole fraction. The equilibrium model predicted exceedingly high values of CO and H₂ in the primary zone, with maximum of 9% and 16% respectively whereas the laminar flamelet model performed much better with values closer to the measured levels of CO, 3%, and H₂, 2.5%. More detailed discussion about the numerical results and their comparison with experimental data is presented in the next section.

5.5 Discussion

5.5.1 Effects of Different Fuels

The measurements presented in Section 5.4.1 are compared with those measured by Tse (1988) to illustrate the differences caused by a change of fuel from methane (CH₄) to kerosene (C₁₂H₂₆ average). Figure 5.37 shows some of the measurements made in the model combustor with natural gas as fuel at air-fuel ratios of 29 and 50 and a preheat of 200K which can be compared with that of Figures 5.7, 5.8, 5.10 and 5.12 with kerosene fuel and with the same mass flow rate of air and air preheat temperature. The different air-fuel ratios for the two fuels in the take-off and ground-idle conditions actually give rise to the same potential heat release in the two experiments.

The distributions of major species are similar but the high temperature region occupied almost the entire width of the half cross-plane in the primary zone with liquid fuel whereas it is limited to the central part in line with the vaporiser exit with gaseous fuel. The differences in the species concentration distributions obtained in the primary zone with the two fuels stem mainly from the flow characteristics at the vaporiser exit for fuel supply which, with the partly premixed liquid fuel with preheated air prior to entry to the combustion chamber with a local air-fuel ratio of 2, had a mass flow rate about three times that of gaseous fuel, so that it can be expected to have dispersed to a greater extent after impinging on the circular head of the combustor. At the exit plane, the influence of higher mass and momentum flow rates through the vaporiser can still be observed, resulting in more uniformly distributed species concentrations and temperature profiles and hence in an improvement in pattern factor from 0.29 to 0.19 based on the central 80% of the area of the exit plane for the take-off condition.

The effects of the fuel are also illustrated by the difference in the ratio of concentrations of CO and H₂ observed in the primary zone where the breakdown of fuel have not been completed due to insufficient air. With similar concentrations of CO₂ formed in the primary zone in the two cases, the gaseous fuel, with its carbon to hydrogen ratio of 0.25, gave similar concentrations of CO (3%) and H₂ (2.5%) whereas with liquid fuel, with carbon to hydrogen ratio of about 0.5, resulted in a lower concentration of H₂ (2%) than of CO (3.5%).

5.5.2 Performance of the Vaporiser

The performance of the model combustor is dependent on the ability of the vaporiser to atomise the fuel since the droplet size of the fuel spray,

the spray density and the residence time determine the type of combustion that takes place in the combustor as suggested by Tuttle et al (1973). In the present combustor configuration, it is difficult to quantify the residence time because large scale recirculation was found within the flowfield. The characteristics of an airblast atomizer, similar to that used in the present study, have been studied by Lorenzetto and Lefebvre (1977) and a correlation was derived for droplet size, that is

$$\text{SMD} = 0.95 \left(\frac{(\sigma_l W_l)^{0.33}}{V_l \rho_l^{0.37} \rho_a^{0.3}} \right) \left(1 + \frac{W_l}{W_a} \right)^{1.7} + 0.13 \mu_l \left(\frac{D}{\sigma_l \rho_l} \right)^{0.5} \left(1 + \frac{W_l}{W_a} \right)^{1.7} \quad (5.2)$$

which represents the Sauter mean diameter and is in good agreement with that given by Nukiyama and Tanasawa (1939) who performed the first major study of airblast atomiser. The operating conditions of the atomiser in the present experiment were within the range for which equation (5.2) is applicable except that the air was at ambient condition for the study by Lorenzetto and Lefebvre (1977). The mean droplet diameter given by the T-vaporiser, estimated from equation (5.2), is around 60 μm at 295K. A 200K preheat would further decrease the droplet size due to evaporation and the experiments by Rizkalla and Lefebvre (1975) suggest an inverse relationship between air temperature and droplet size. Hence, most of the droplets coming out of the exit of the T-vaporiser would be below 50 μm . Moreover, larger droplets were unable to follow the flow of air in the vaporiser through a 180° turn and would hit on the inside wall of the vaporiser to further atomise and evaporate so that the droplet size of the liquid fuel emerging from the vaporiser to the chamber was further reduced.

There are three possible ways for liquid fuel to burn, as suggested by Nicholls et al (1980): (i) with adequate supply of premixed air and sufficiently small drops, a premixed gaseous fuel-air flame will be formed; (ii) with sufficiently large drops, single droplet combustion will result and (iii) the fuel spray will burn as a turbulent non-premixed gaseous flame as proposed by Chigier (1974) and by Onuma and Ogasawara(1974). Considering the close values of concentrations of UHC and temperatures obtained with the two fuels in the primary zone, the third model would be appropriate for describing the combustion processes in the primary zone of the combustor. This is also supported by the finding of Attya and Whitelaw (1984) who stated that for droplet diameter less than $50\mu\text{m}$, and increasing with preheat temperature, droplet models are unnecessary to account for the combustion processes.

5.5.3 Characterization of Combustion Process by Mean Flow Properties and Equilibrium Reaction

The representation of combustion process is essential to the modelling of combustor flow in gas turbine combustor and the extent to which it can be represented by mean flow properties and equilibrium reaction can be quantified by comparing the measured species concentrations and equilibrium concentrations. The accuracy of inferring temperature from enthalpy balance can also be assessed with respect to the results presented in Section 5.4.1.

For fuel-lean conditions, the equilibrium assumption can give values of major species concentrations to within 7% of the measured values as in the case of air-fuel ratio of 43 at the exit plane, where the equivalence ratios were less than 0.4, as shown in Figures 5.7 and 5.8. As the values of equivalence ratio increased, the discrepancies between values of measured

and equilibrium concentrations also increased as observed in the case of the air-fuel ratio of 26 at the exit plane. For fuel-rich conditions, as in the over stoichiometric region in the primary zone, the equilibrium concentrations of CO and H₂ were overpredicted by an order of magnitude. With the co-existence of fuel and air, this implies that the chemical process for fuel breakdown was kinetically controlled rather than mixing controlled. Hence, for a better description of the combustion process, the specification of more than one scalar variable is necessary. The effects of fluctuations, through the construction of a probability density function of the single scalar variable, has been studied by Jones and Toral (1983) and large differences were also observed in fuel-rich regions. This necessitates the description of combustion with more variables such as the two scalar probability density function model of Aguirre-Saldivar (1986).

The differences between the enthalpy balance and measured temperatures have been studied in detail by Toral and Whitelaw (1982), Jones and Toral (1983) and Tse (1988). The relation between enthalpy balance temperature, which is density-weighted, and the unweighted measured temperature is given by,

$$T_h = T_m + \frac{\overline{\rho T'}}{\overline{\rho}} + \frac{\Sigma(C_{p\alpha} \overline{m_\alpha T''})}{\Sigma(C_{p\alpha} \overline{m_\alpha})} \quad (5.3)$$

with T_h always lower than T_m according to Tse (1988). The enthalpy balance temperature for the air-fuel ratio of 43 at the exit, Figure 5.38, can be compared with the measured values of Figure 5.7. The two temperature distributions are close to each other with maximum difference of less than 70K. Assuming a 10% radiation loss, the measured temperatures are actually about 100K higher which is in accordance to the finding of Tse

(1988). It can also be concluded that the second and last terms on the right hand side of Equation (5.3) are of the order of radiation loss.

5.5.4 Discrepancies between Measurements and Predictions in Isothermal Flow

The numerical results presented in Section 5.4.2 (a) are compared with those obtained by Tse (1988) to quantify the differences between measurements and prediction. Figures 5.39 and 5.40 show comparisons of velocity profiles in primary zone and at the exit of the combustor respectively. The maximum v-velocity predicted at a location 10mm above the primary jet exit is about 12% lower than that measured because the air jets after leaving the plunged holes tend to neck with maximum velocities roughly 6mm above the jet exit, as suggested by Tse (1988), so that centreline velocities increased to about 1.15 times that of the exit velocities. This physical phenomenon cannot be reproduced in the calculation. At 20mm above the jet exit, the difference in values of maximum velocity can still be observed. The extra momentum of the jets, giving larger velocity gradients in the vertical jet plane, increased the velocity of the fluid between the jets and led to the discrepancy observed in Figure 5.39b. The u-velocity at the same location was well predicted given that the boundary conditions for all the inlet ports were estimated.

At the exit, measurements and predictions give similar profiles with a difference of about 25% in the maximum velocity. The positions of maximum velocity are at $z=-14$ and 42mm corresponding to the dilution jets impingement regions. As mentioned in Section 5.4.2 (a), the flow patterns in those regions and further downstream depend on the upper jet entry angle which is small but finite. The difference, therefore, is attributed to the incorrect representation of the jet entry angle and the

value of which, in fact, may not be constant but vary with time in the model combustor.

Figure 5.41 allows comparison of passive scalar distributions with differences less than 5% for the region downstream of primary jets. Further upstream, the calculated values were dependent on the boundary condition of the vaporiser exit and, due to the lack of measurement of passive scalar near the vaporiser exit, the error associated with the passive scalar distribution cannot be quantified.

5.5.5 Discrepancies between Measurements and Prediction in Combusting Flow

The measurements of temperature and species concentrations of the model Gem combustor at an air-fuel ratio of 29 are shown in Figures 5.42 and 5.43 which are taken from Tse (1988) and can be compared with the result of the simulations with the equilibrium model, Figures 5.32 and 5.33, and with the laminar flamelet model, Figures 5.34 and 5.35. Predicted results show very little mixing between fuel and air in the primary zone, as found in the combusting flow calculations in combustors by Jones and McGuirk (1979) and Wilhelmi (1984), resulting a region of high equivalence ratios at the bottom half of the primary zone whereas the experimental data suggest that the fuel was carried upward by the row of primary jets and entering the primary zone vortex where combustion took place near the top wall of the combustor. The distribution of the calculated turbulent viscosity is shown in Figure 5.44 and in general, is greater than the molecular viscosity by less than a factor of 100. This is unrealistically low and, according to Tennekes and Lumley (1972), the ratio between the turbulence and molecular viscosity is of the order of the local Reynolds number which is greater than around 1000 with an estimated characteristic

length scale of 1mm. The calculated heat release through fuel breakdown in the bottom half of the primary zone was also higher, due to the assumption of fast chemistry, where the measurements suggest a partly kinetically controlled combustion. This resulted in greater momentum flux predicted for the fluid in the lower half of the combustor and hence, instead of being carried upward by the primary jets, the high temperature fluid passed around the air jets to flow straight to the dilution zone. A combustion model with an element of chemical-kinetics control is needed to represent the combustion process with greater accuracy.

5.6 Conclusions

This chapter presents experimental results of the model annular combustor with liquid fuel at an air preheat temperature of 515K and with two air-fuel ratio simulating the take-off and ground-idle conditions. Results of numerical simulations in the same configuration have also been presented and encompass non-reacting flow with a comprehensive parametric study of the influence of inlet parameters. Combusting flow calculations were also carried out with two combustion models to study the effect of combustion and the merits of the models. The main findings are summarised in the following paragraphs:

1. The change of fuel from natural gas to kerosene caused a decrease in pattern factor from 0.29 to 0.19 due to better mixing resulting from the premixing of fuel and air in the vaporiser and impingement of the mixture on the head of the combustor.

2. An increase in the air-fuel ratio from 26 to 43, which corresponded to a reduction of the primary zone air-fuel ratio from fuel rich to lean, resulted in an increase in pattern factor from 0.19 to 0.37 and in combustion efficiency of from 94.2% to 96.2%.
3. The enthalpy balance temperatures, calculated from measured species concentrations, were close to the measured temperature which implies that the correlations of temperature fluctuations is of the same order as the radiation loss of the thermocouple under the operating conditions of the present experiments.
4. The equilibrium calculations, based on the measured equivalence ratios, led to exit plane distributions of the concentrations of major species which were in good accord with the measurements but it failed to account for the reaction processes in the primary zone.
6. Numerical experiments suggested that the strength of the primary zone vortex was independent of the flow split through different air inlet ports within a reasonable range and the mixing process within the primary zone was determined by the amount of air within it.
7. In the dilution zone, the nature of the jet impingement region was sensitive to the inlet angle for the top dilution jets. An inclination of 5° to the vertical resulted in an impingement without slip, but angles in excess of 10° led to no impingement. Also, the introduction of a non-uniform velocity profile at the vaporiser exit led to a maximum mixture fraction of less than 30% in the centre plane compared with over 70% for a uniform profile. This high level of sensitivity of the predicted flowfield to the inlet

boundary conditions indicated that measurements or empirical correlations of flow splits and jet angles are important to ensure accurate predictions.

8. For reacting flow calculations, combustion increased the forward and backward velocity maxima in the primary recirculation zone from 12 to 30m/s. The equilibrium model resulted in high primary zone concentrations of CO and H₂, with maximum predicted values of 9% and 16% respectively, compared with 2.5% and 1.75% of the measurements. The flamelet model gave a better representation with maximum values of 4.5% for CO and 3.5% for H₂.

9. Differences in fuel-air mixing process were observed in reacting flow calculations due to the inability of the predictions to simulate the correct entrainment of fuel rich fluid into the upstream edges of the primary jets and to be carried upwards to feed the primary zone vortex. This casts doubt on the precision of the turbulence and scalar transport model when applied to combustor flow calculations.

Table 5.1 Geometric Details of Gem Combustor

Cooling Slots	Slot Height/mm	No. of Holes	Diameter/mm	Pitch/mm
2	1.65	21	2.25	5.91
3	1.38	24	2.00	5.20
4	1.38	24	1.65	5.20
5	1.60	18	2.40	6.84
7	1.10	18	1.65	6.84

Primary and Dilution Holes	Hole Diameter/mm	No. of Hole	Pitch/mm
1	6.2	4	28
6	4.6	5	28
8	4.6	2	56

Table 5.2 Properties of Aviation Grade Kerosene

Property	Value
Chemical Formula	$C_{12}H_{26}$ average
% Composition by Mass	
Carbon	85.8
Hydrogen	14.1
Sulphur	0.1
Density at 1 atm and 15.6°C (kg/m ³)	780.0
Boiling Point (°C)	255.0
Mean Specific Heat Capacity between 0 - 100°C (kJ/kg K)	2.09
Calorific Value at 15°C (MJ/kg)	
Gross	46.5
Net	43.5
Stoichiometric Air Requirement (Mass of Dry Air/Mass of Fuel)	14.96

Table 5.3 Air Distributions in Gem-60 Combustor

	% of Mass Flow
Cooling Slots	
2	8.2
3	12.2
4	14.8
5	8.2
7	6.2
Primary and Dilution Holes	
1	27.1
6	15.8
8	7.5

Table 5.4 Performance of the Model Combustor

	Air-Fuel Ratio	Pattern Factor	El _{CO} (g/kg fuel)	El _{UHC} (g/kg fuel)	Combustion Efficiency (%)
Kerosene	26	0.19*	200	22	92.3
	43	0.37	77	10	96.2
Methane	29	0.29	181	21	94.2
	50	0.48	64	11	97.6

*Based on enthalpy balance temperature

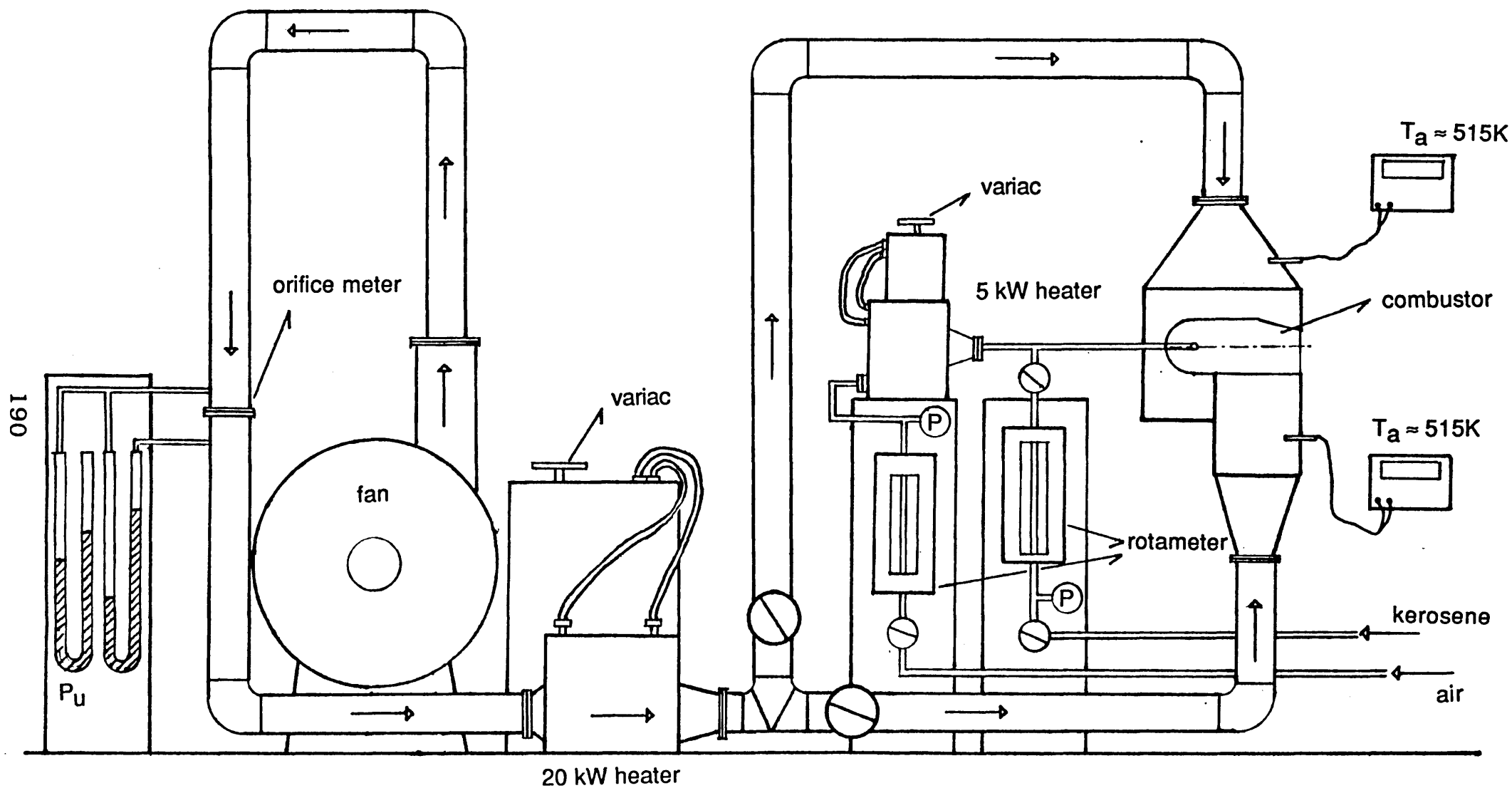


Figure 5.1 Flow Arrangement

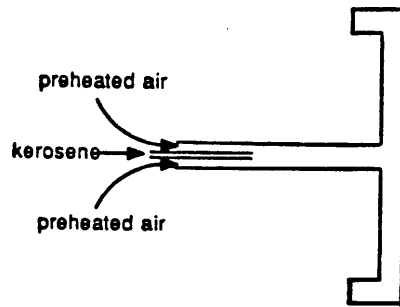


Figure 5.2 Details of Vaporiser

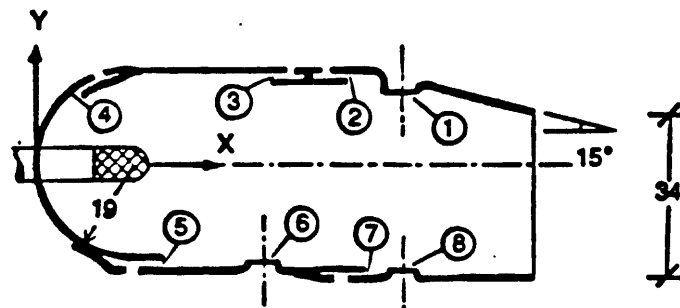
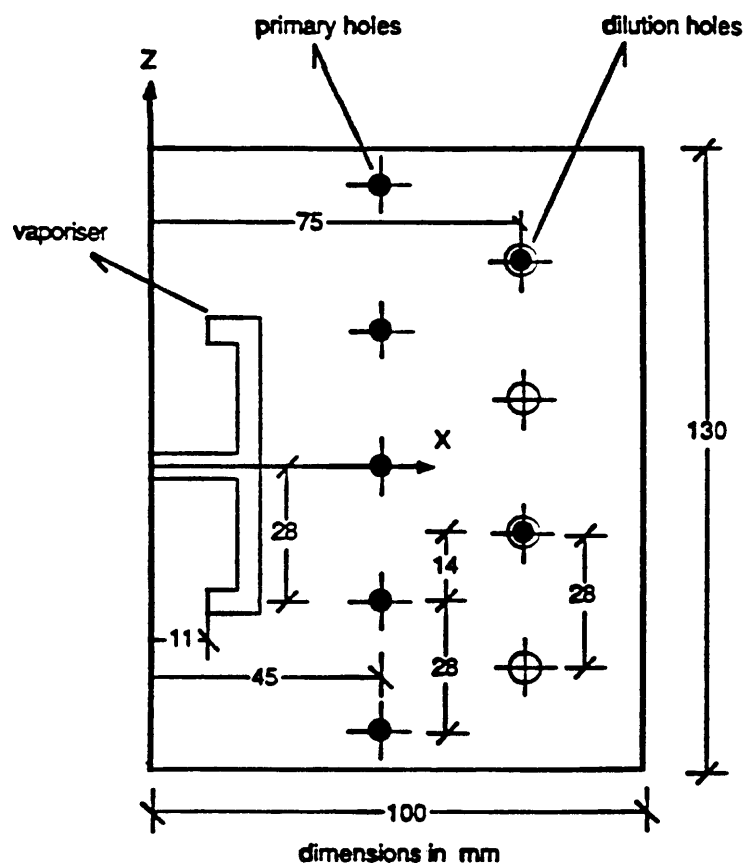


Figure 5.3 Combustor Geometry

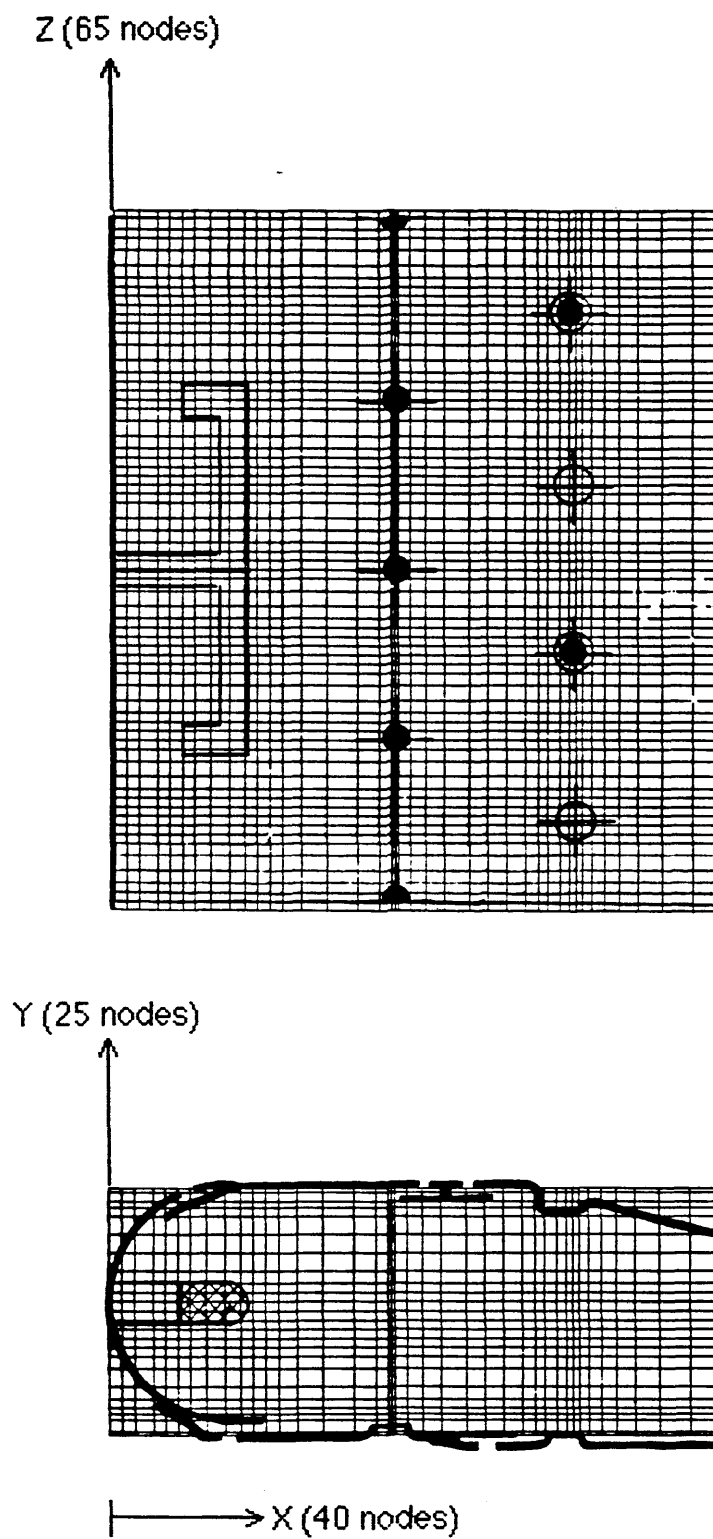
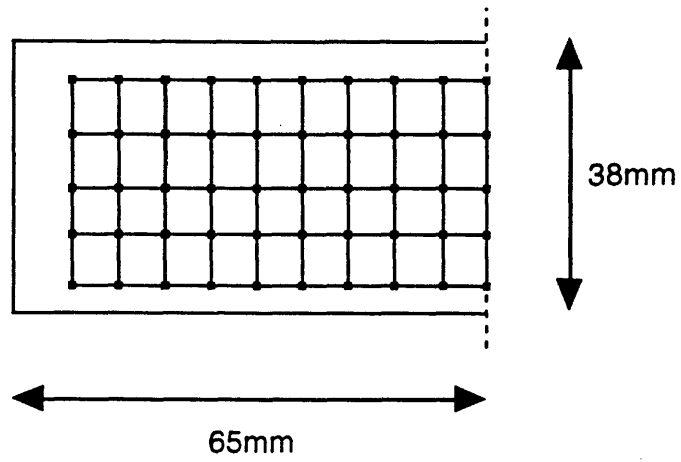


Figure 5.4 Finite Difference Mesh used in the Calculations

(a)



(b)

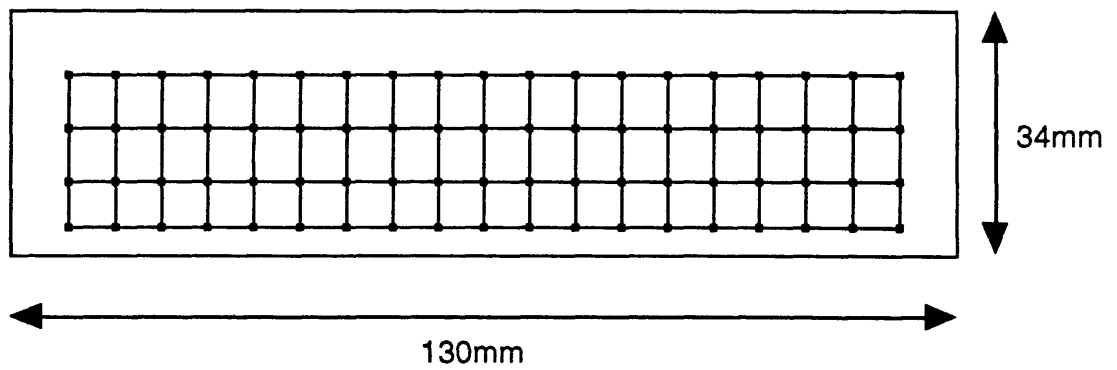


Figure 5.5 Measurement Grids for the Combustor
(a) Primary Zone
(b) Exit Plane

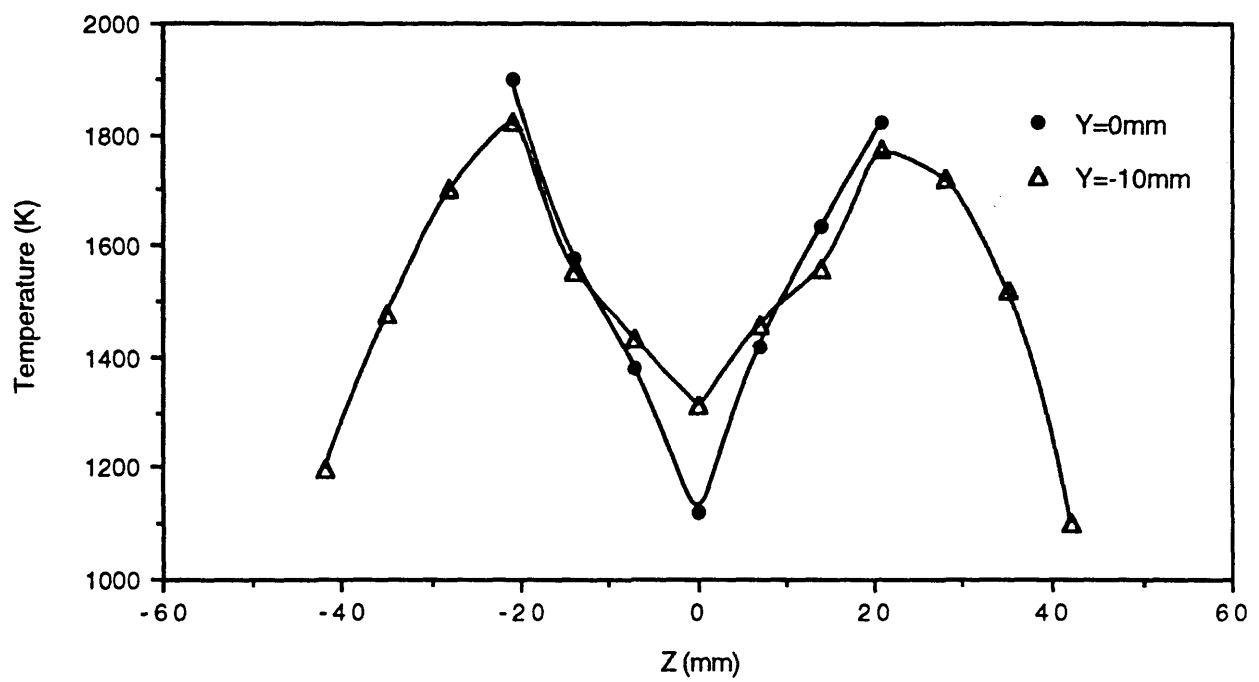


Figure 5.6 Mean Temperature Measurement at x=36mm

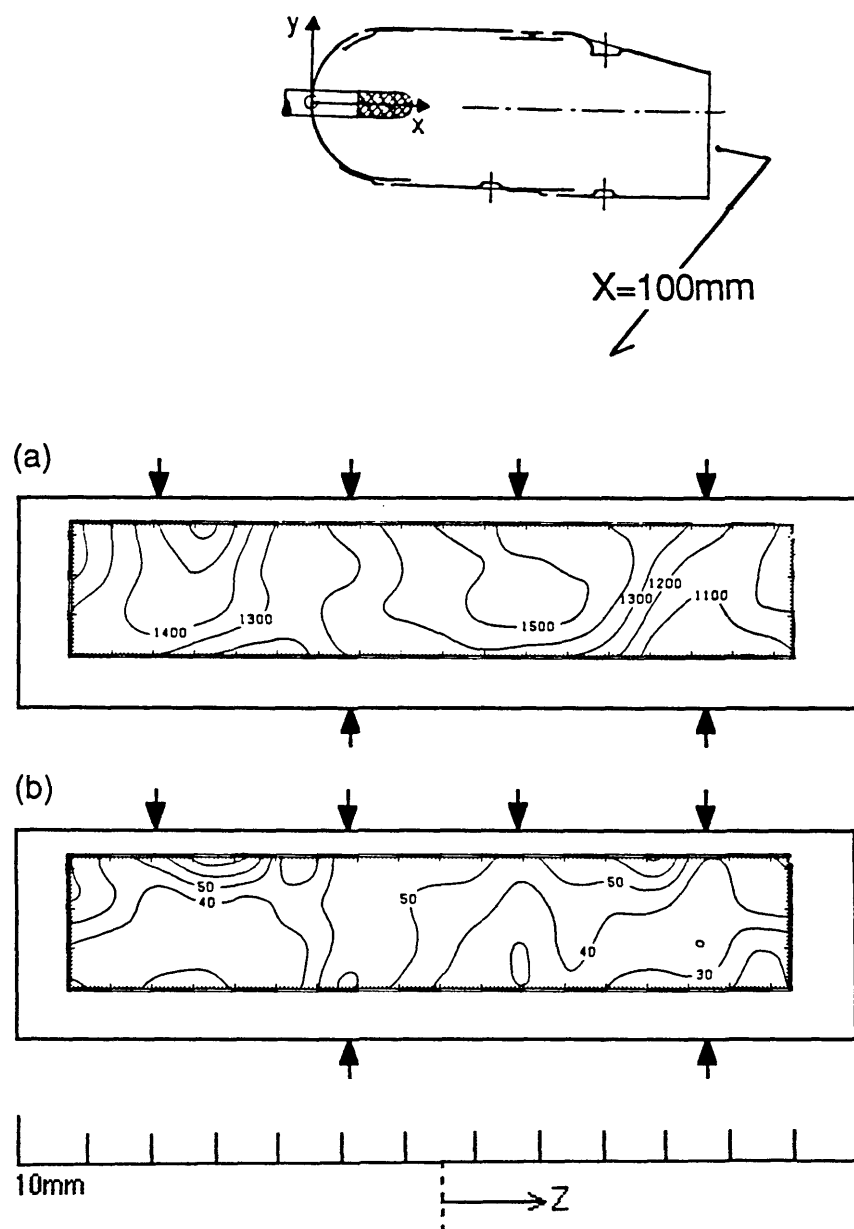


Figure 5.7 Measured Temperature Distributions in Exit Plane for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=43$

(a) Mean Temperature

(b) rms Temperature Fluctuation

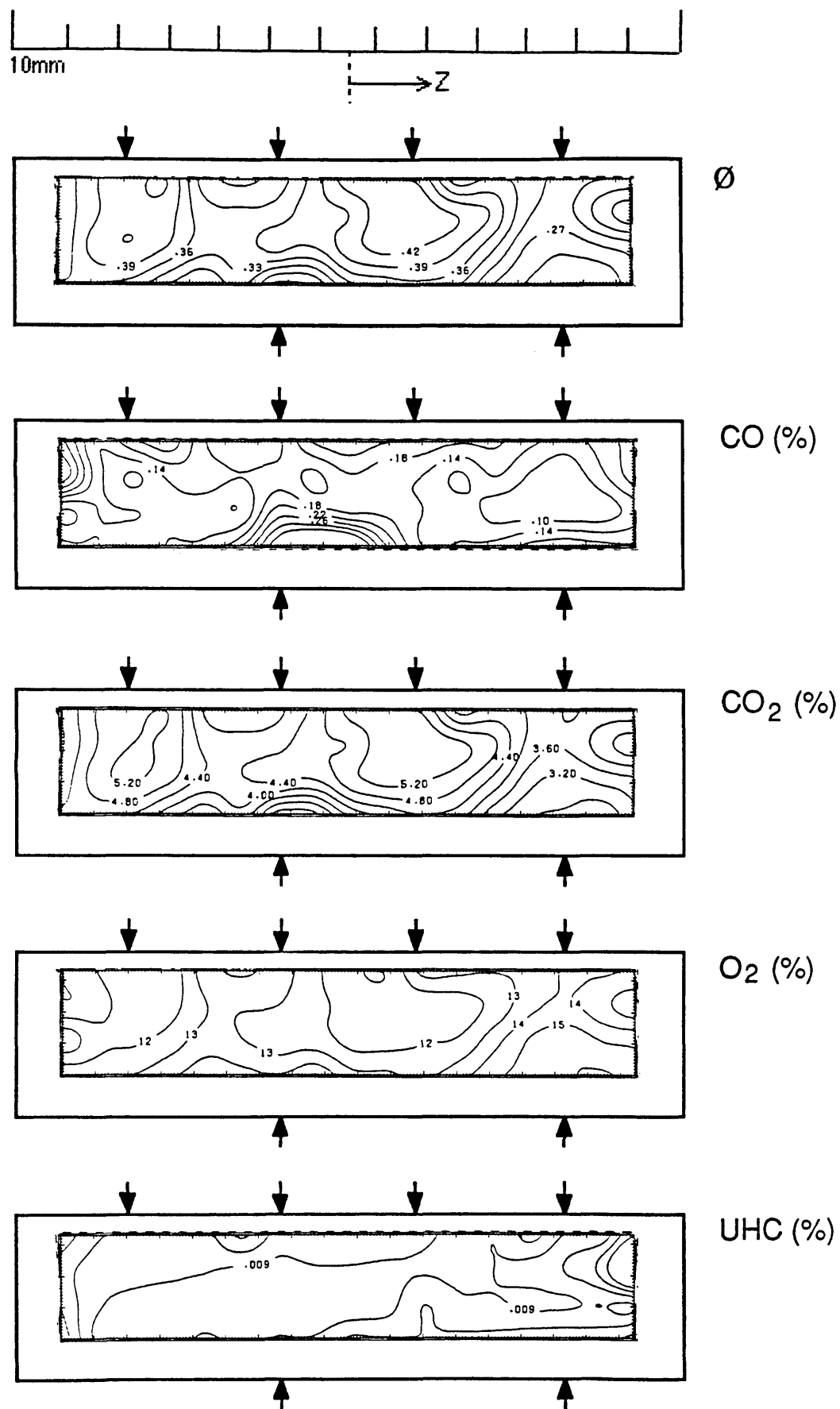


Figure 5.8 Distributions of Measured Species Concentrations in Exit Plane for $T_{inlet}=515K$ and $AFR=43$

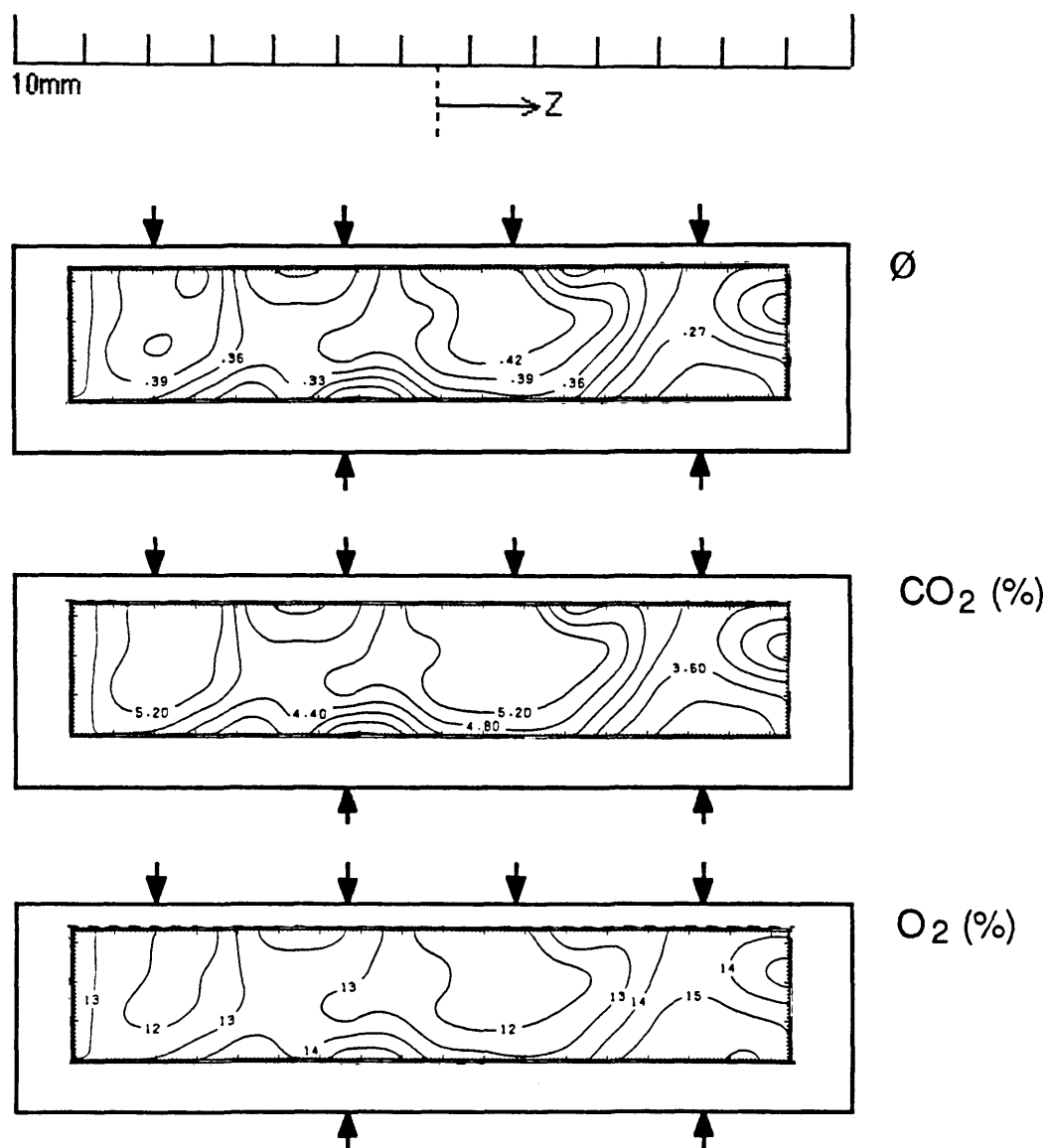


Figure 5.9 Distributions of Calculated Equilibrium Concentrations in Exit Plane for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=43$

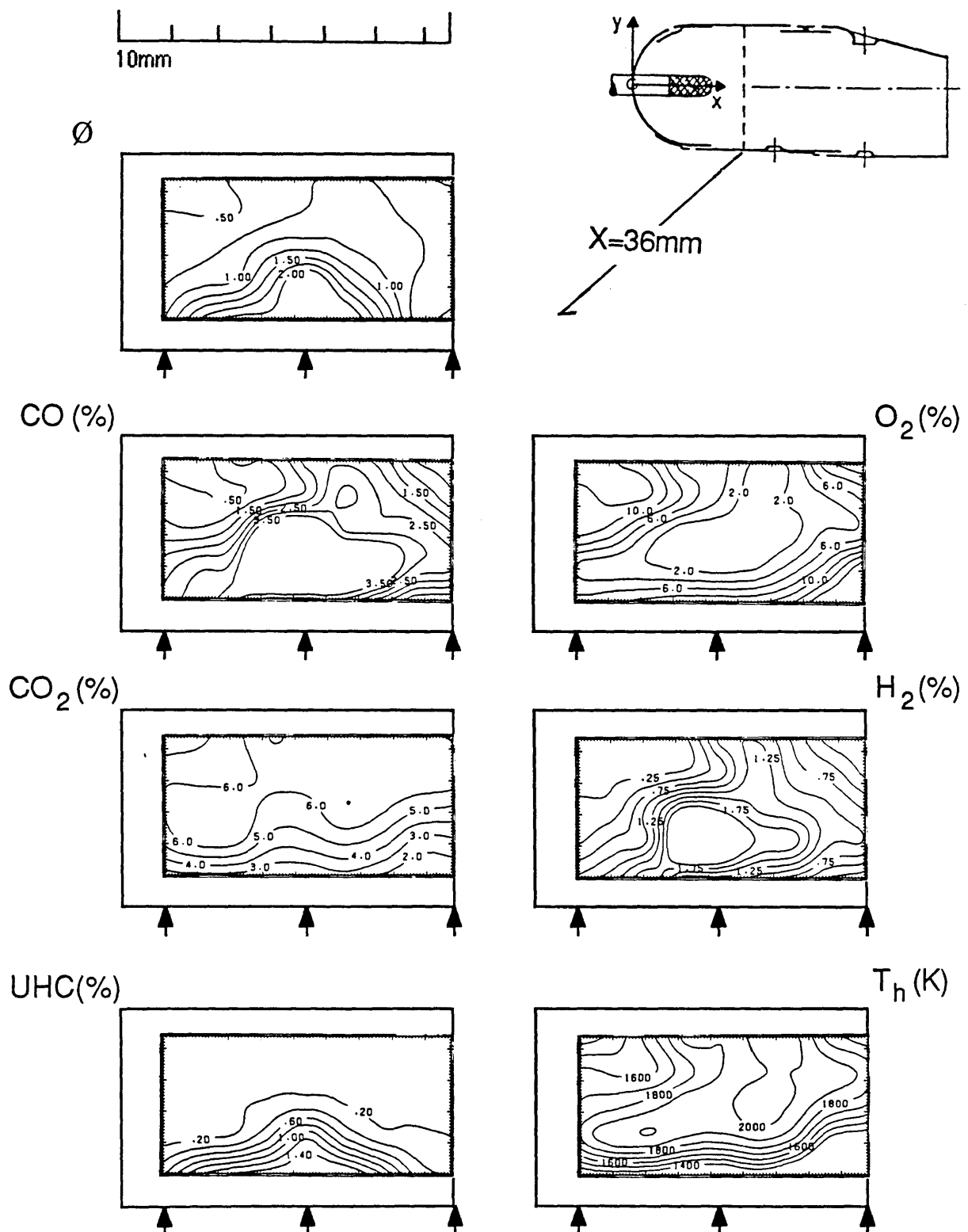


Figure 5.10 Distributions of Measured Species Concentrations and Calculated Enthalpy Balance Temperature in Primary Zone for $T_{inlet}=515K$ and $AFR=26$

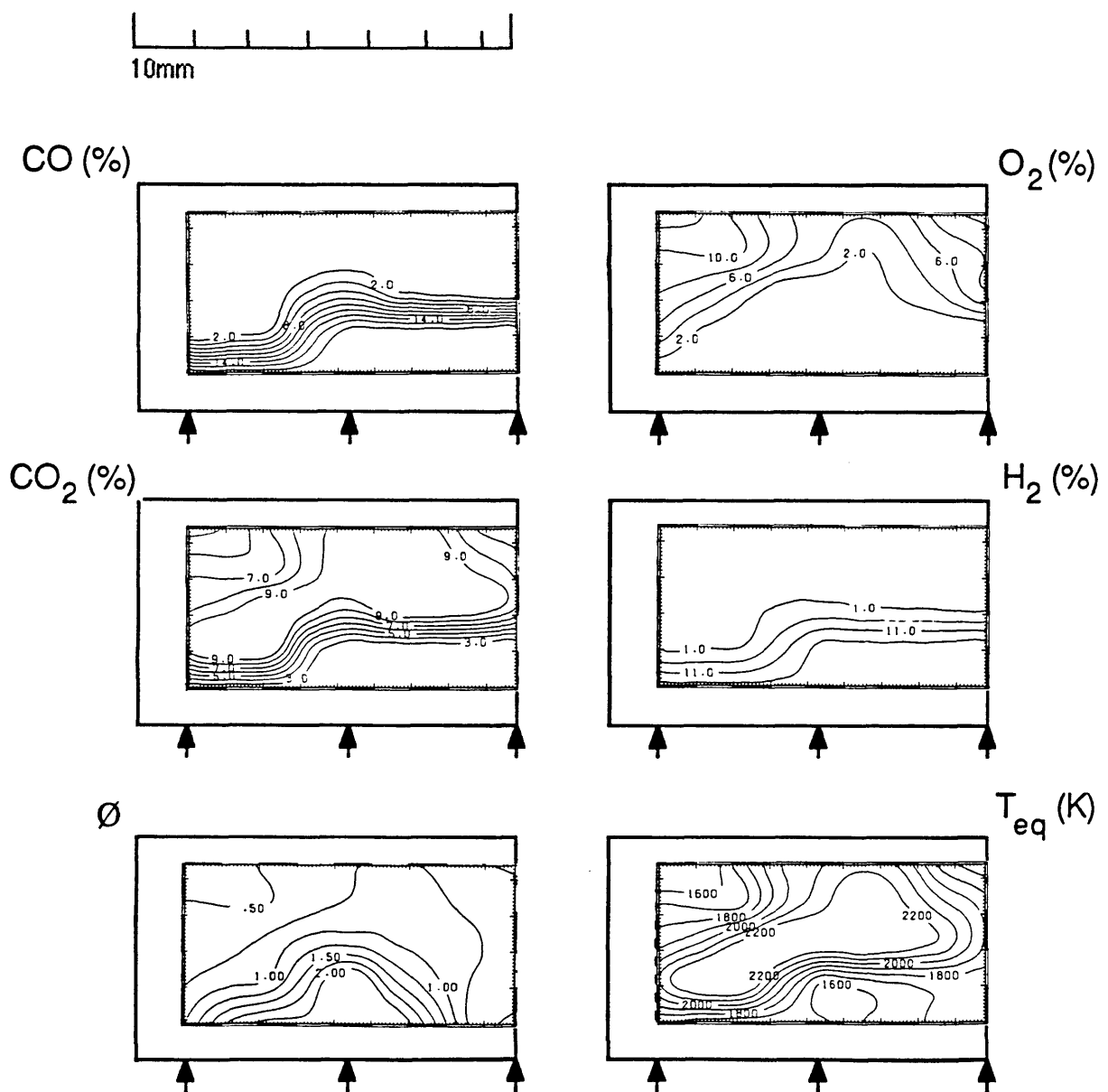


Figure 5.11 Distributions of Calculated Equilibrium Concentrations and Temperature in Primary Zone for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=26$

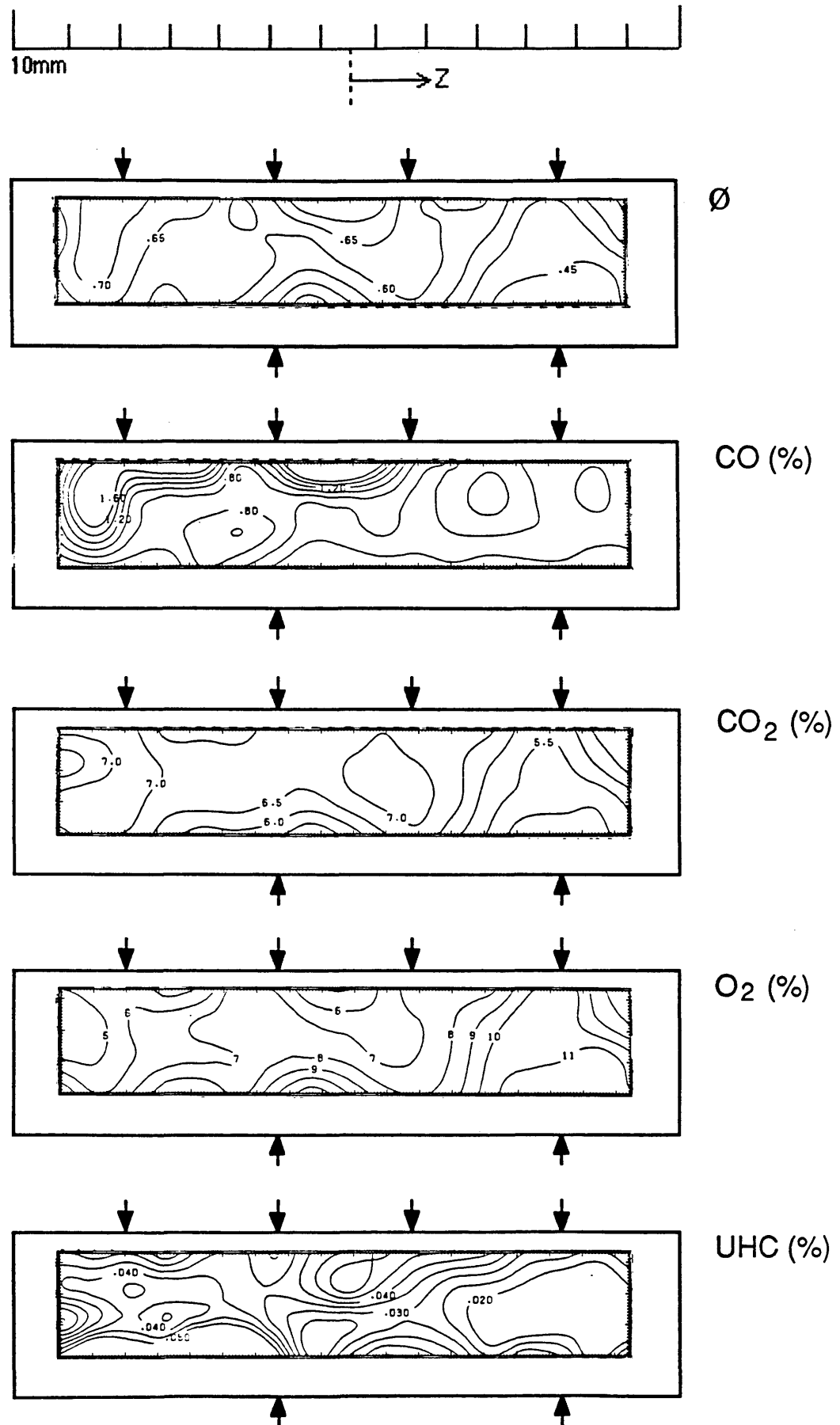


Figure 5.12 Distributions of Measured Species Concentrations in Exit Plane for $T_{inlet}=515K$ and $AFR=26$

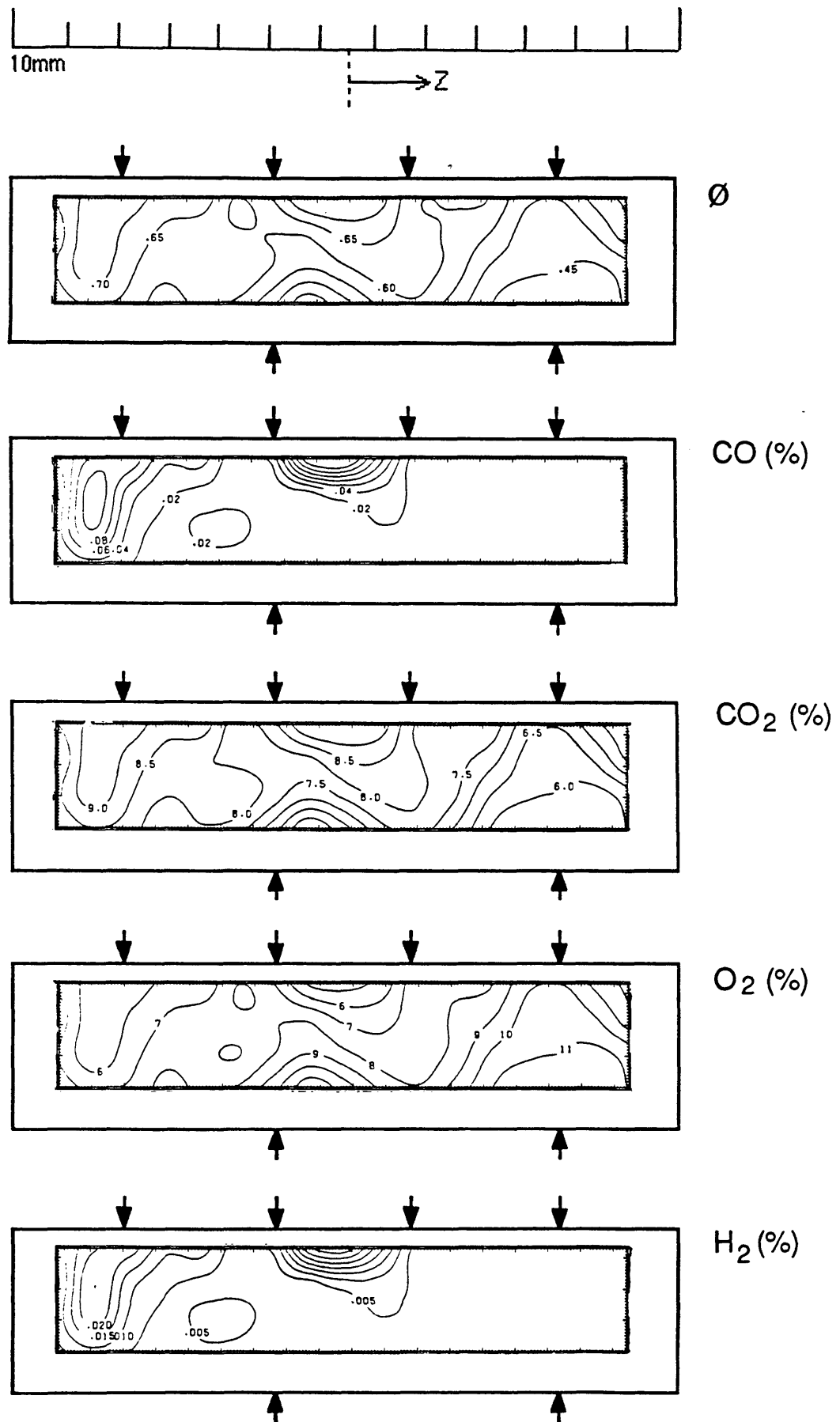


Figure 5.13 Distributions of Calculated Equilibrium Concentrations in Exit Plane
for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=26$

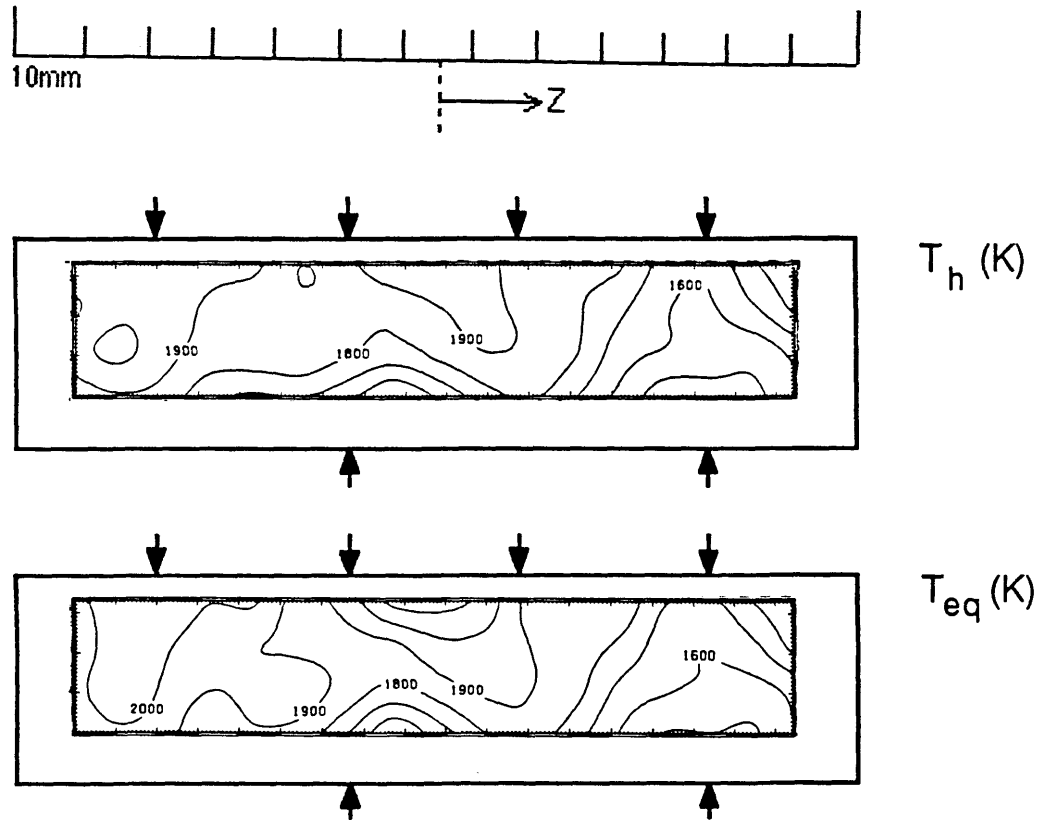


Figure 5.14 Calculated Equilibrium and Enthalpy Balance Temperature Distributions in Exit Plane for $T_{inlet}=515K$ and $AFR=26$

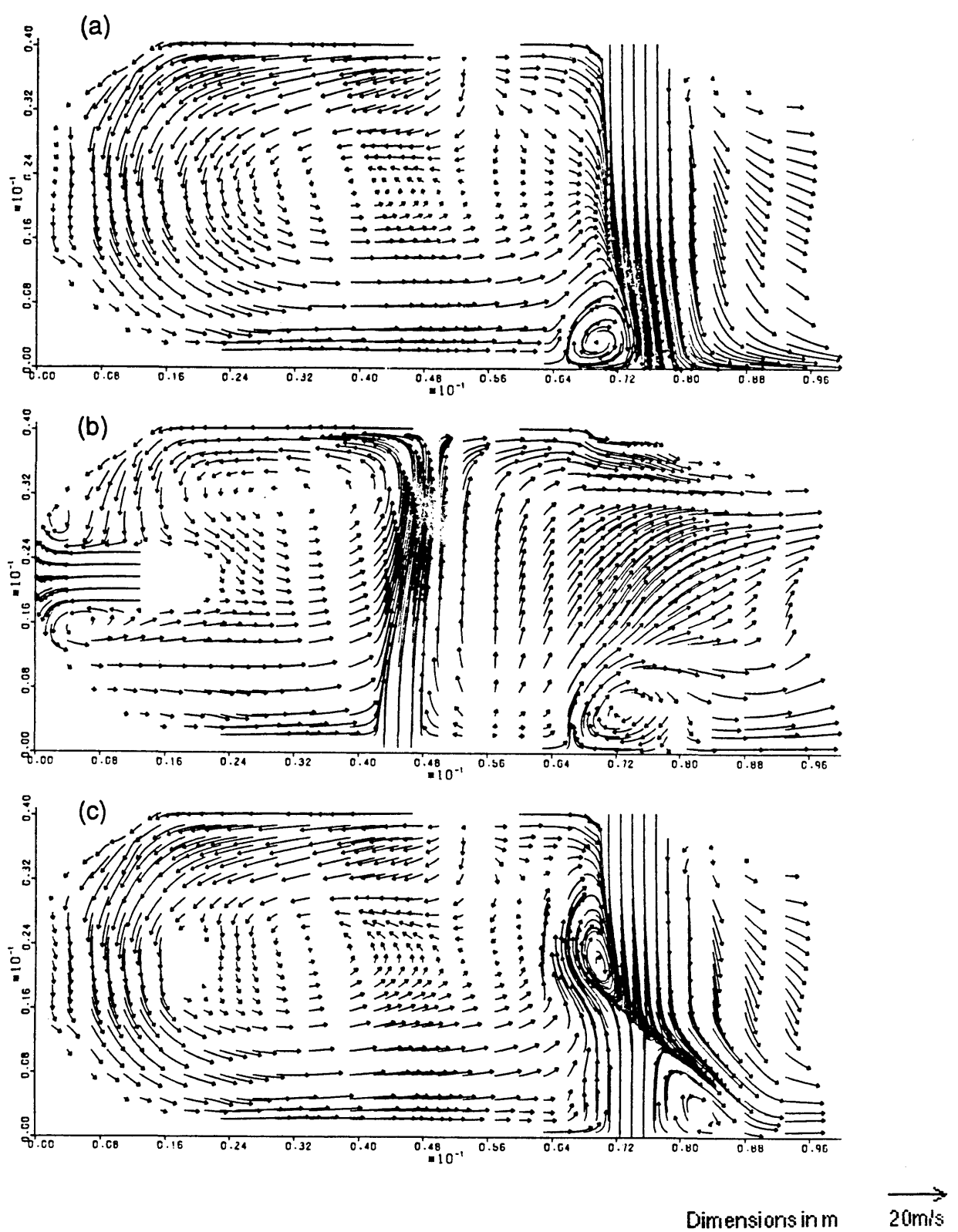
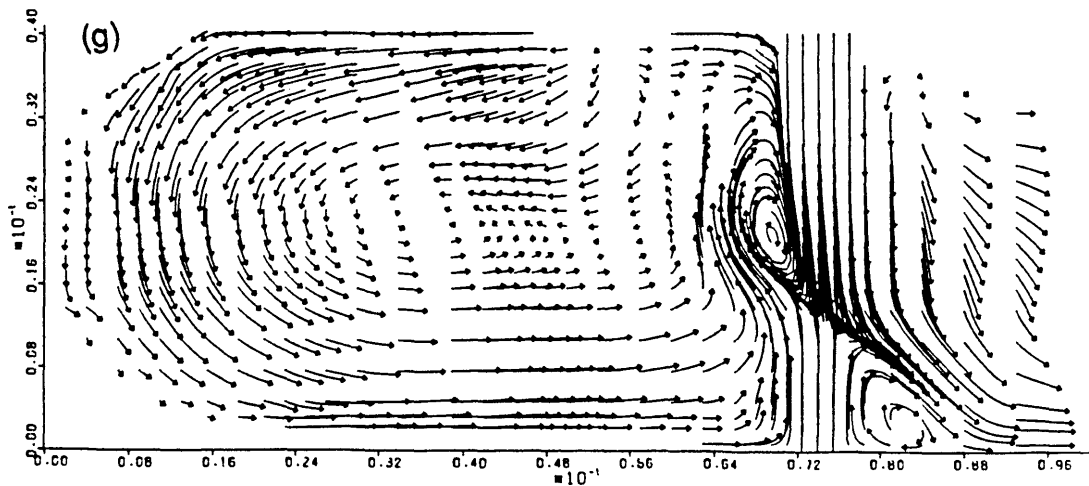
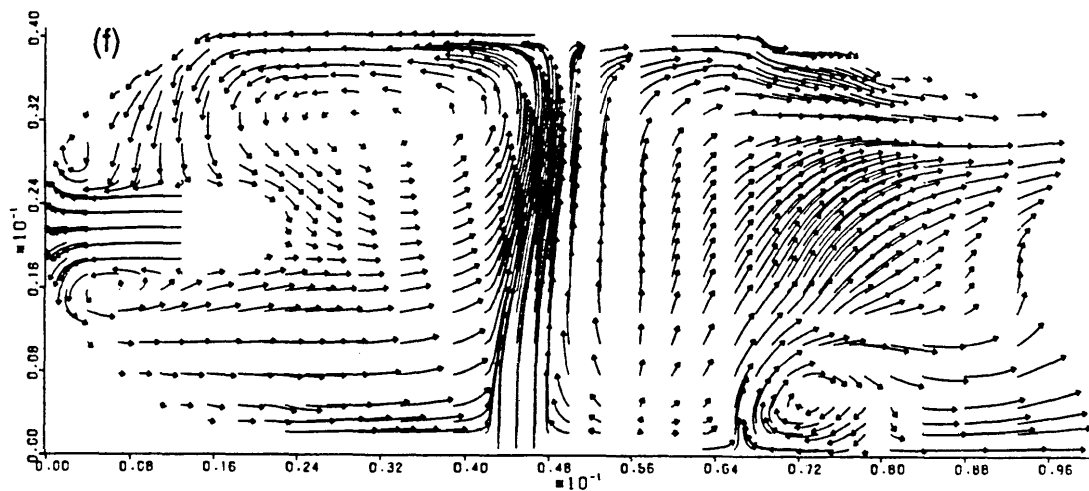
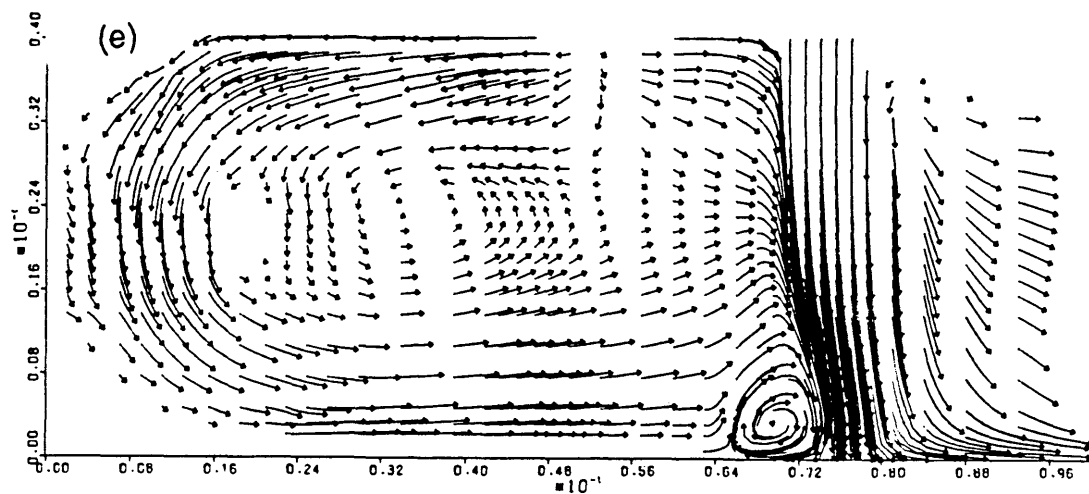
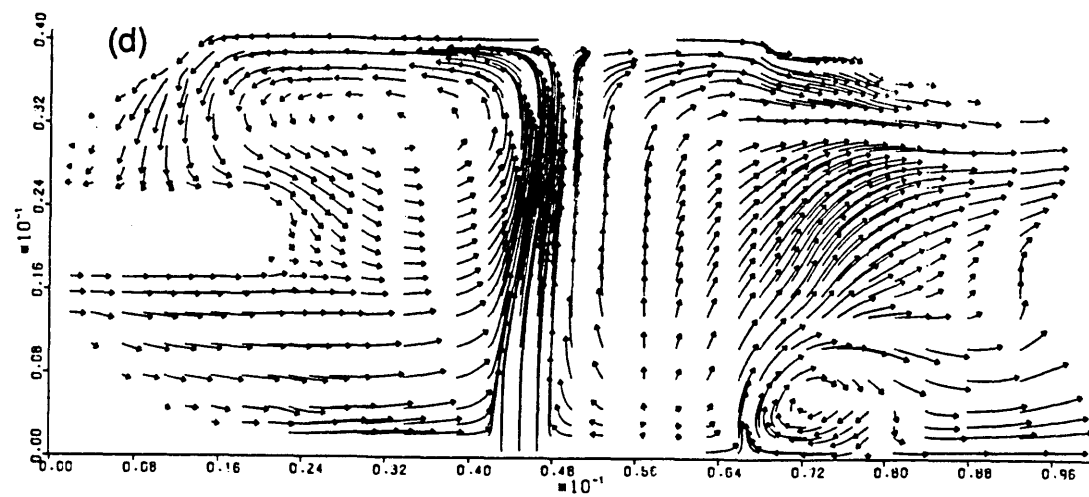


Figure 5.15 Streakline Plot of Isothermal Flow of Combustor

- | | | |
|------------------------|------------------------|------------------------|
| (a) $z = -42\text{mm}$ | (b) $z = -28\text{mm}$ | (c) $z = -14\text{mm}$ |
| (d) $z = 0\text{mm}$ | (e) $z = 14\text{mm}$ | (f) $z = 28\text{mm}$ |
| (g) $z = 42\text{mm}$ | | |



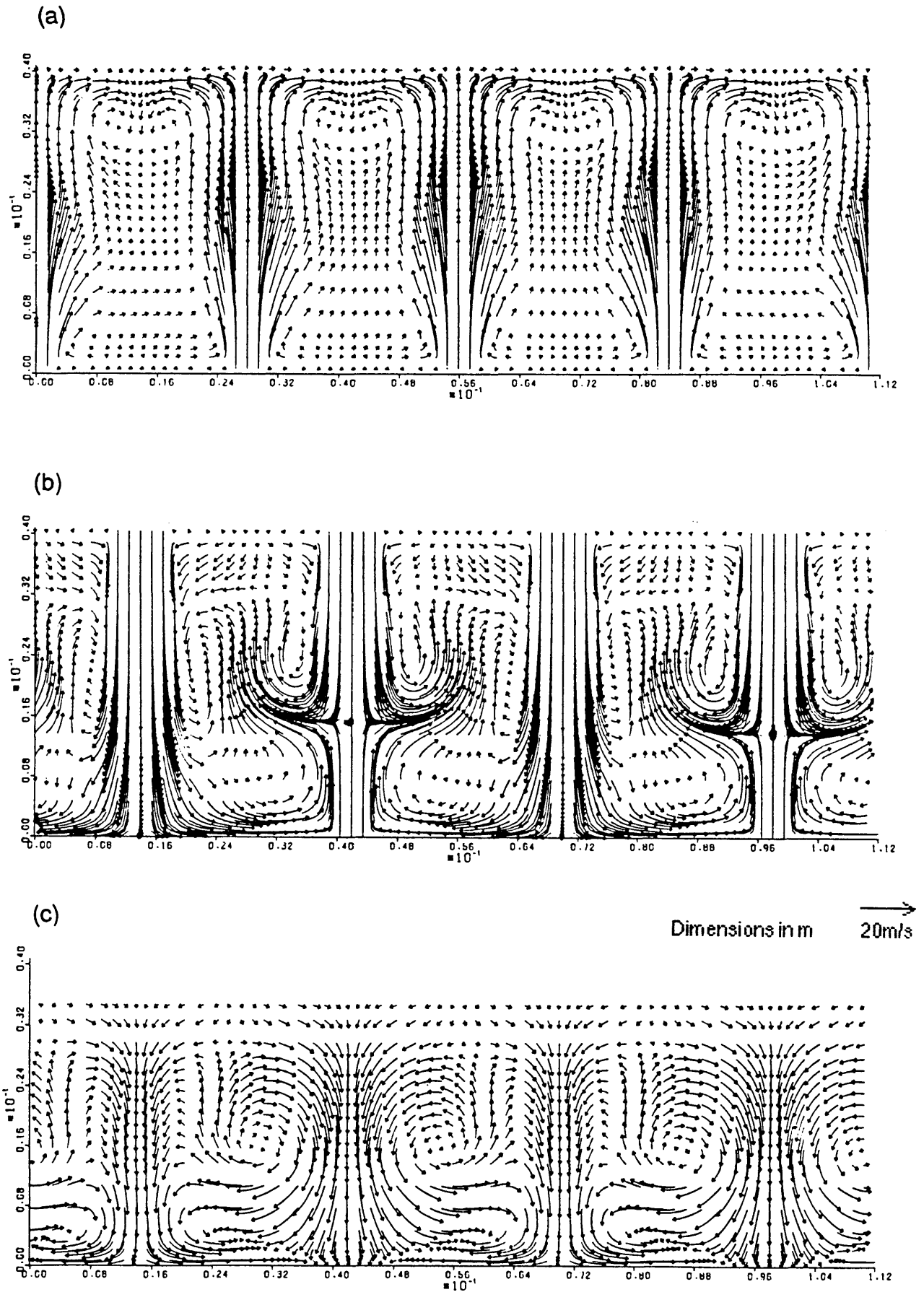


Figure 5.16 Streakline Plot of Isothermal Flow of Combustor in Cross-Stream

Planes

(a) $x=45\text{mm}$

(b) $x=74\text{mm}$

(c) $x=100\text{mm}$

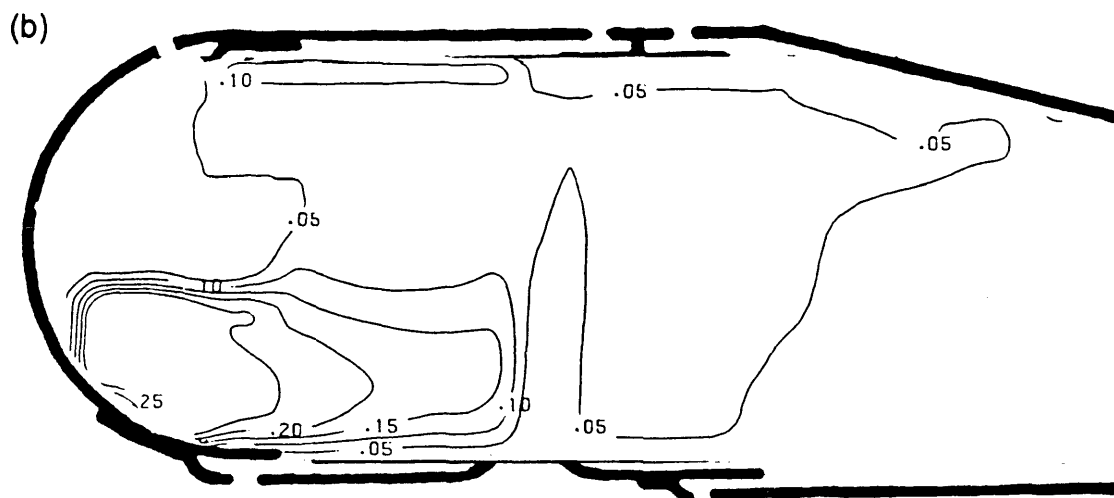
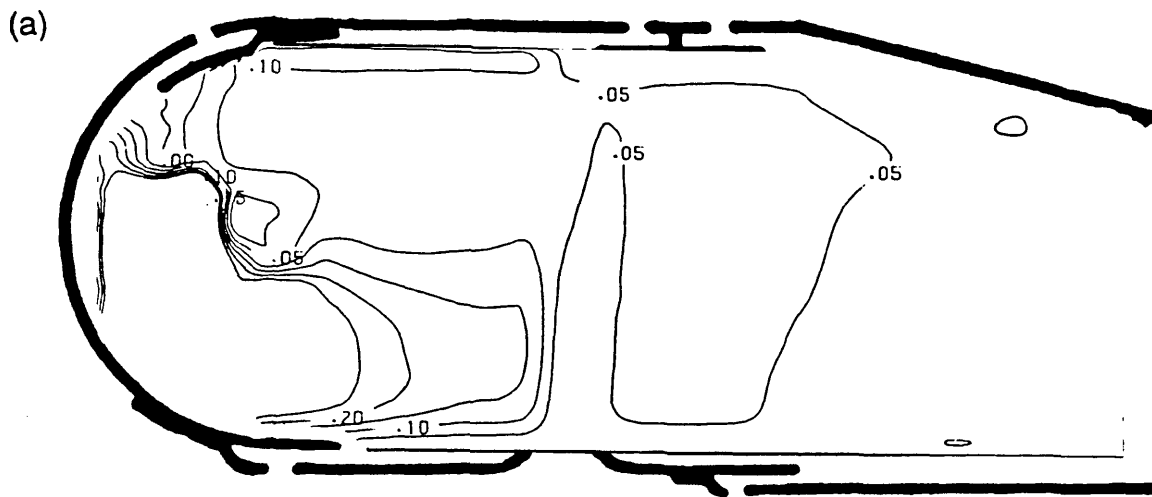


Figure 5.17 Predicted Passive Scalar Distributions in Isothermal Flow
(a) $z=28\text{mm}$ (b) $z=0\text{mm}$

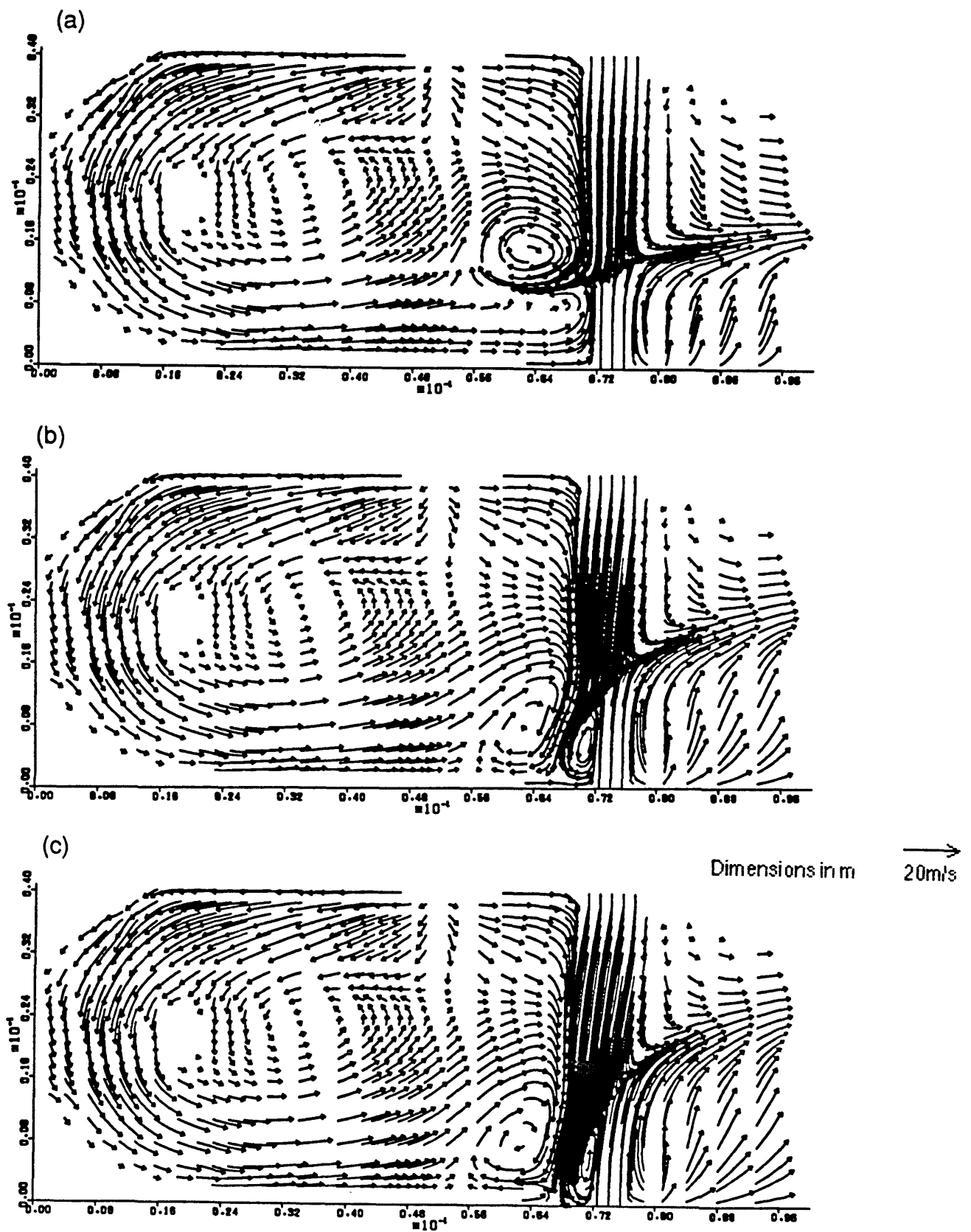


Figure 5.18 The Influence of Dilution Jet Entry Angle to Jet Impingement Region in

Isothermal Flow

(a) $\theta=5.0^\circ$

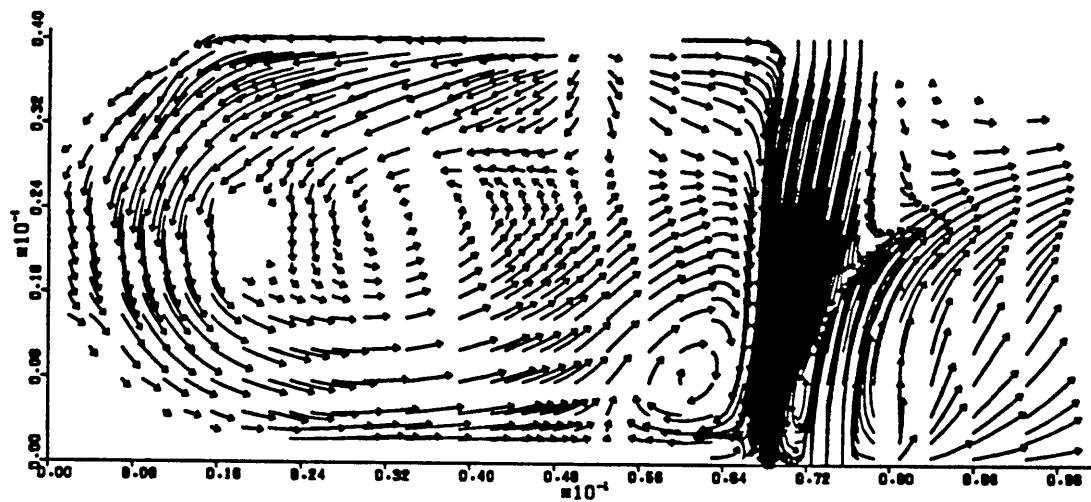
(b) $\theta=7.5^\circ$

(c) $\theta=10.0^\circ$

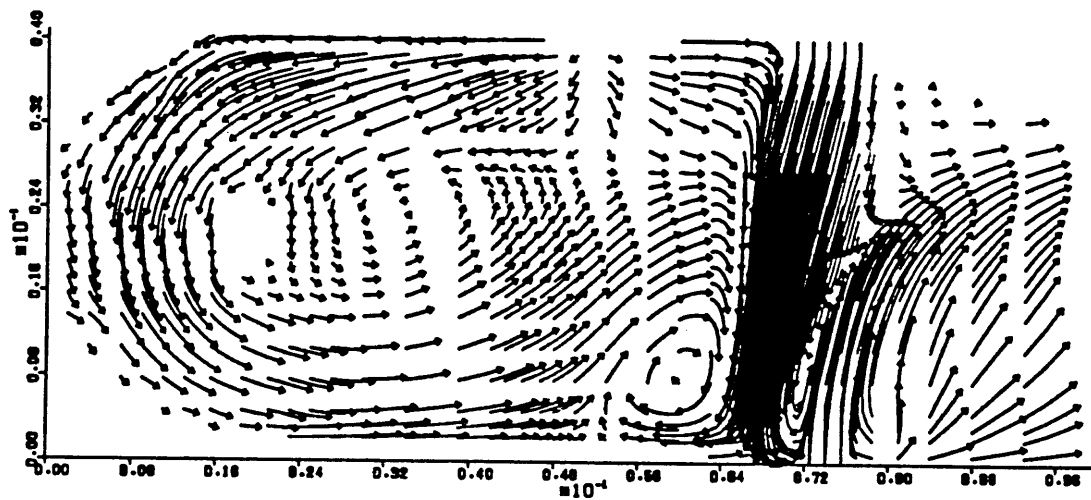
(d) $\theta=12.5^\circ$

(e) $\theta=15.0^\circ$

(d)



(e)



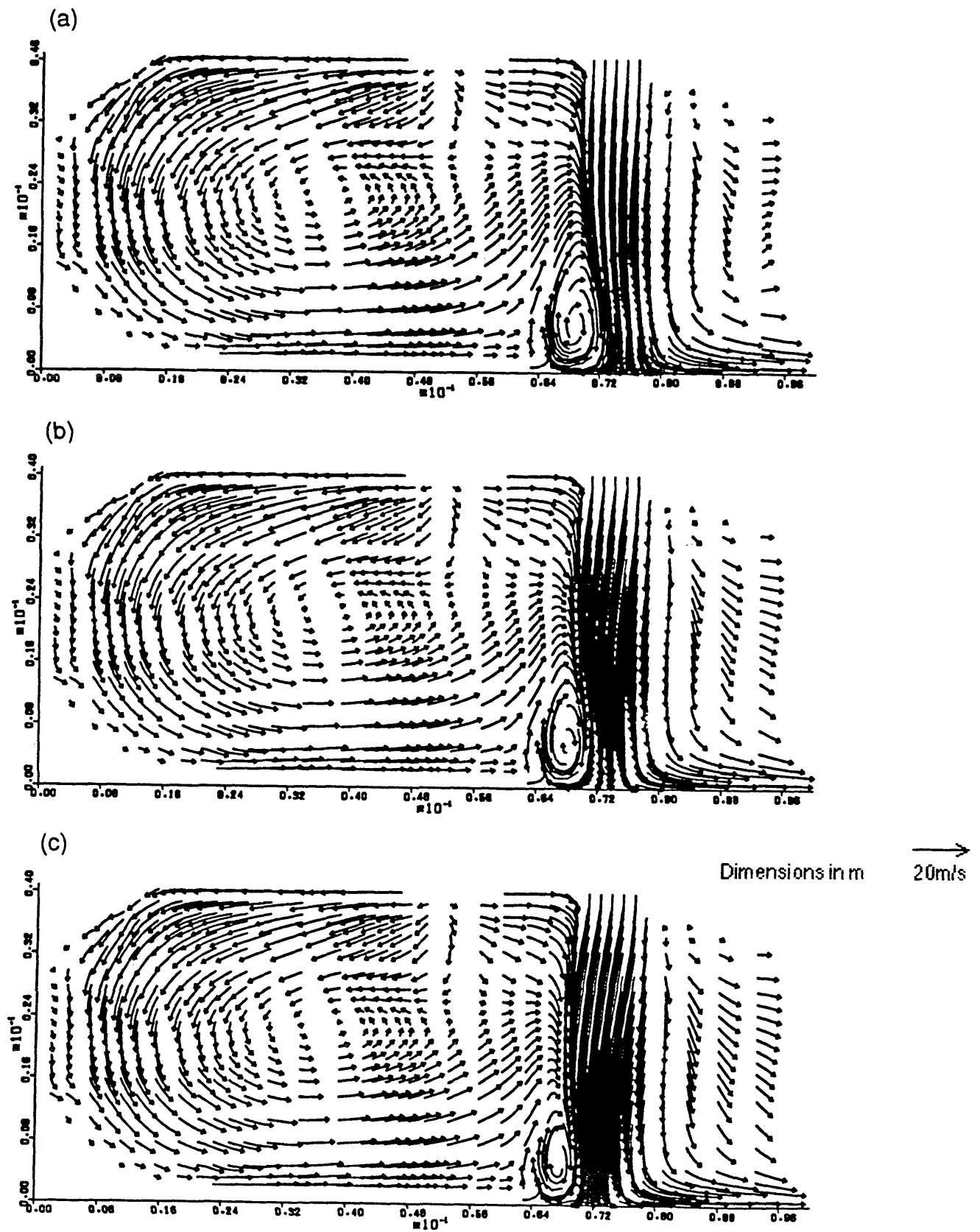


Figure 5.19 Streakline Plot at $z=-42\text{mm}$ for Different Dilution Jet Entry Angle in

Isothermal Flow

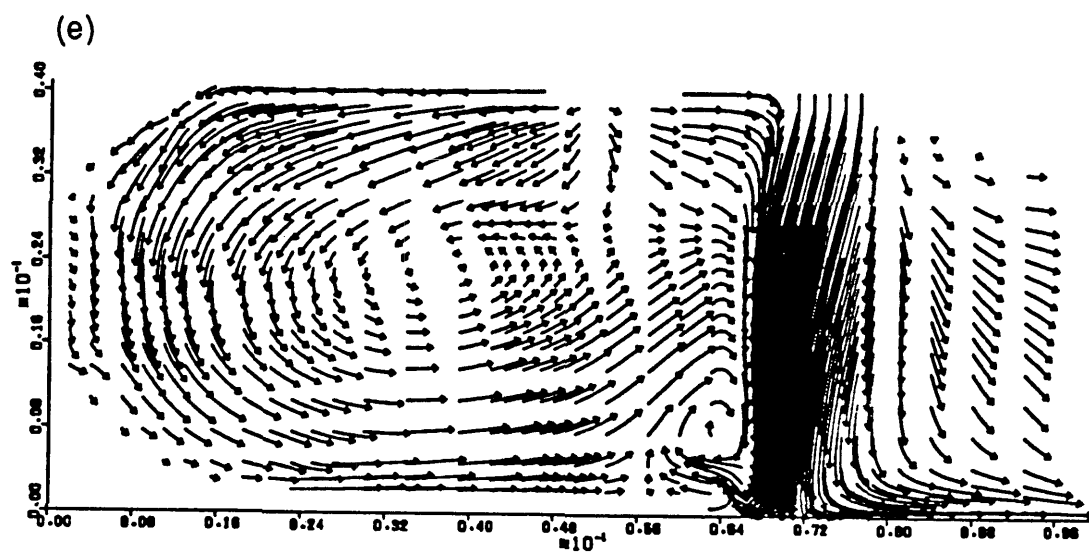
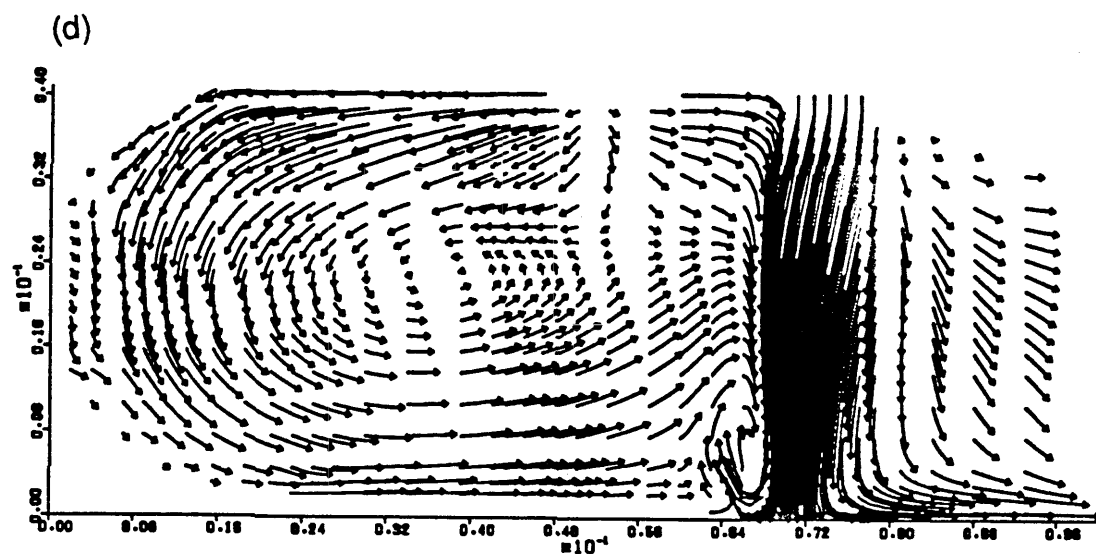
(a) $\theta=5.0^\circ$

(b) $\theta=7.5^\circ$

(c) $\theta=10.0^\circ$

(d) $\theta=12.5^\circ$

(e) $\theta=15.0^\circ$



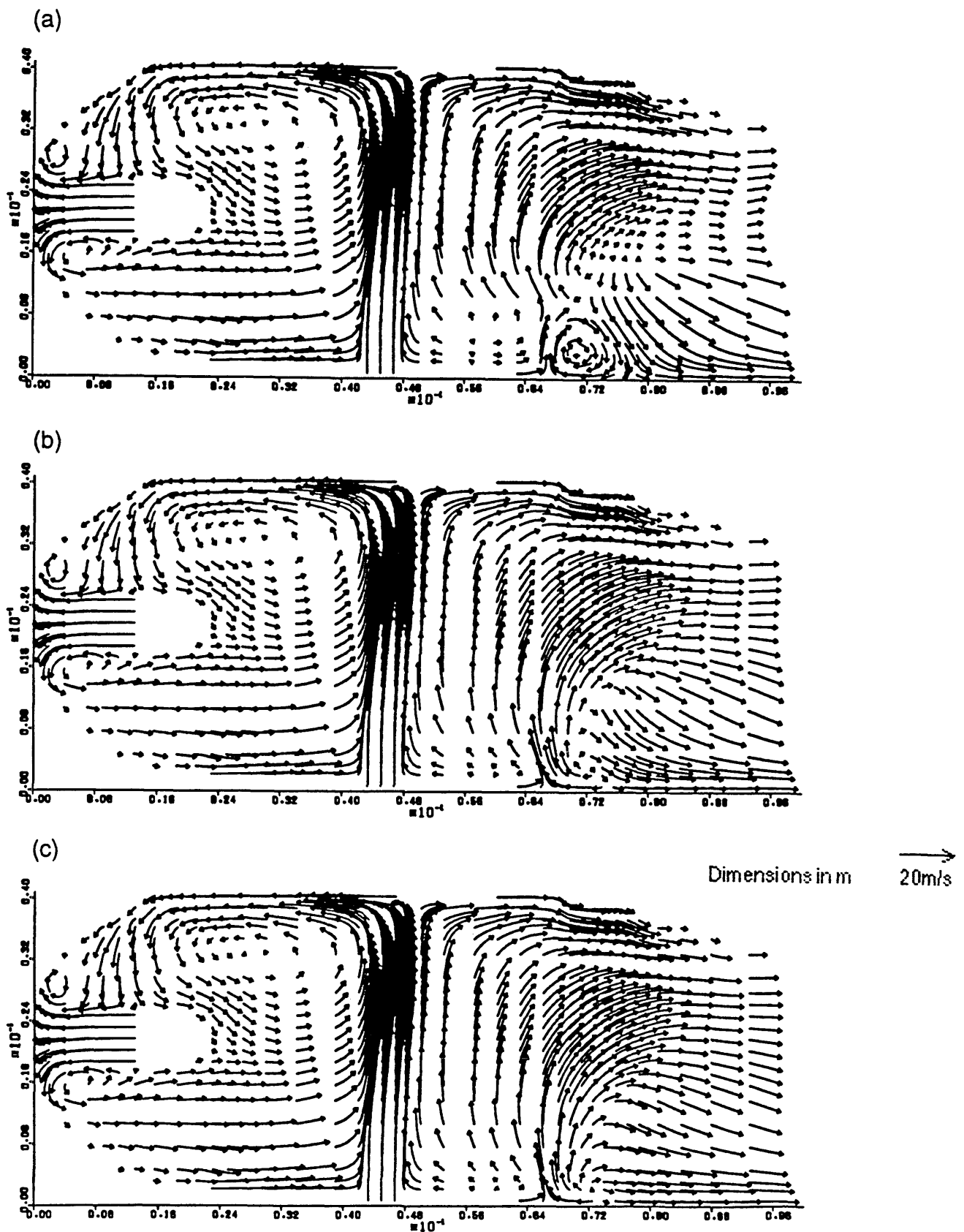


Figure 5.20 Streakline Plot at $z=-28\text{mm}$ for Different Dilution Jet Entry Angle in

Isothermal Flow

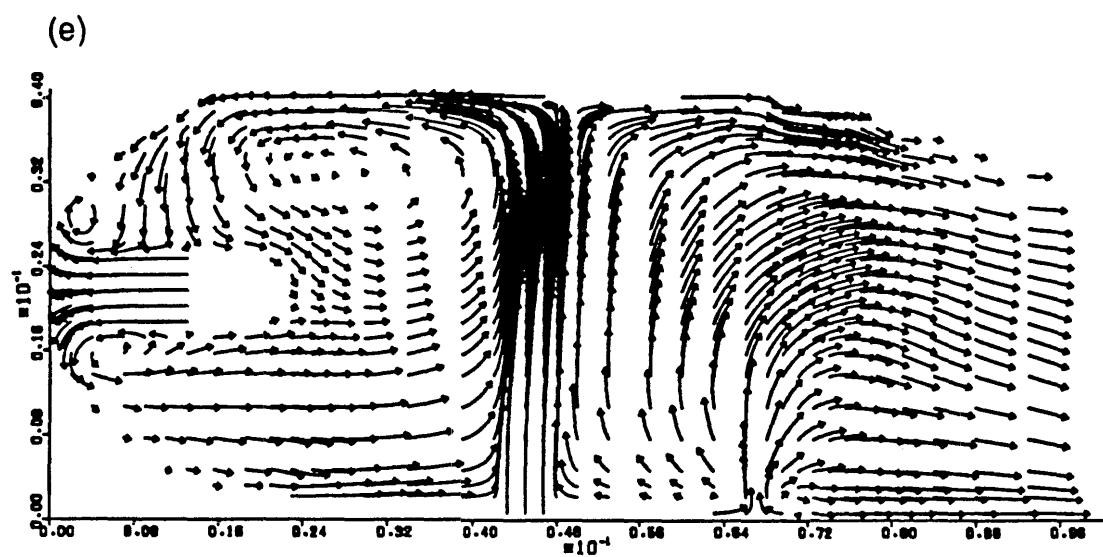
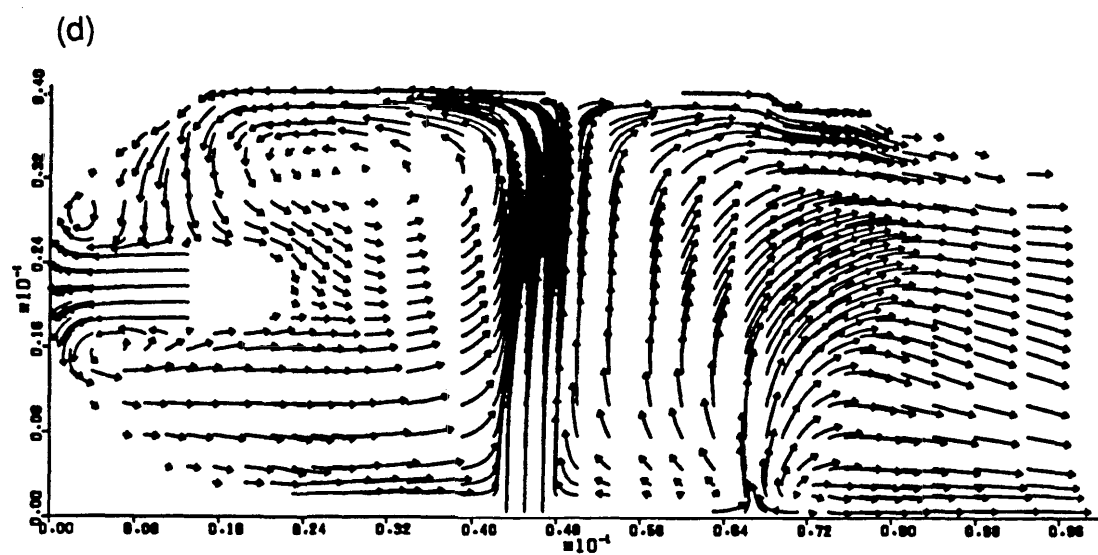
(a) $\theta=5.0^\circ$

(b) $\theta=7.5^\circ$

(c) $\theta=10.0^\circ$

(d) $\theta=12.5^\circ$

(e) $\theta=15.0^\circ$





$z = +28 \text{ mm}$



$z = +14 \text{ mm}$



$z = 0 \text{ mm}$



$z = -14 \text{ mm}$



$z = -28 \text{ mm}$

Figure 5.21 Flow Visualisation of Combustor (Taken from Tse (1988))

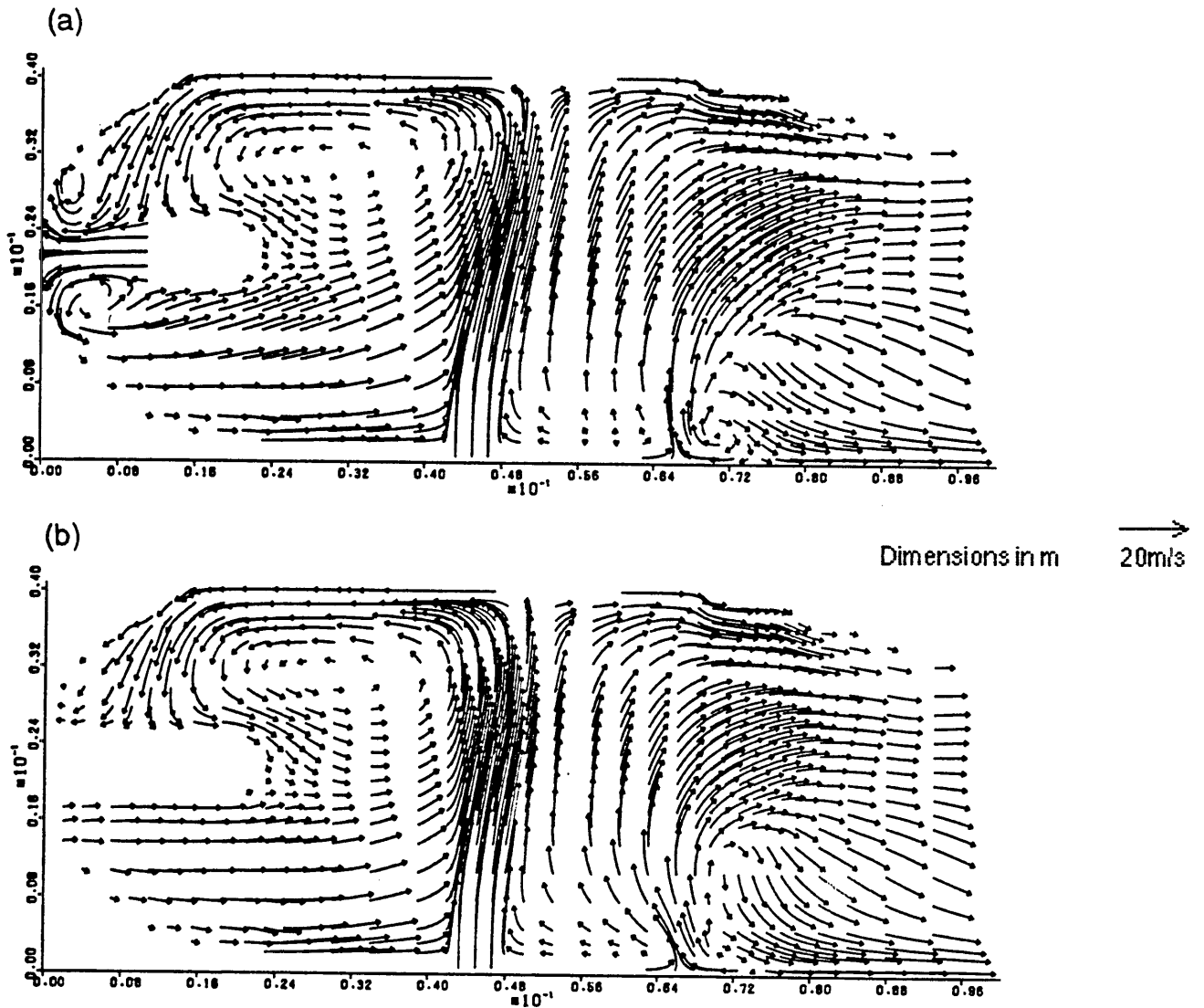
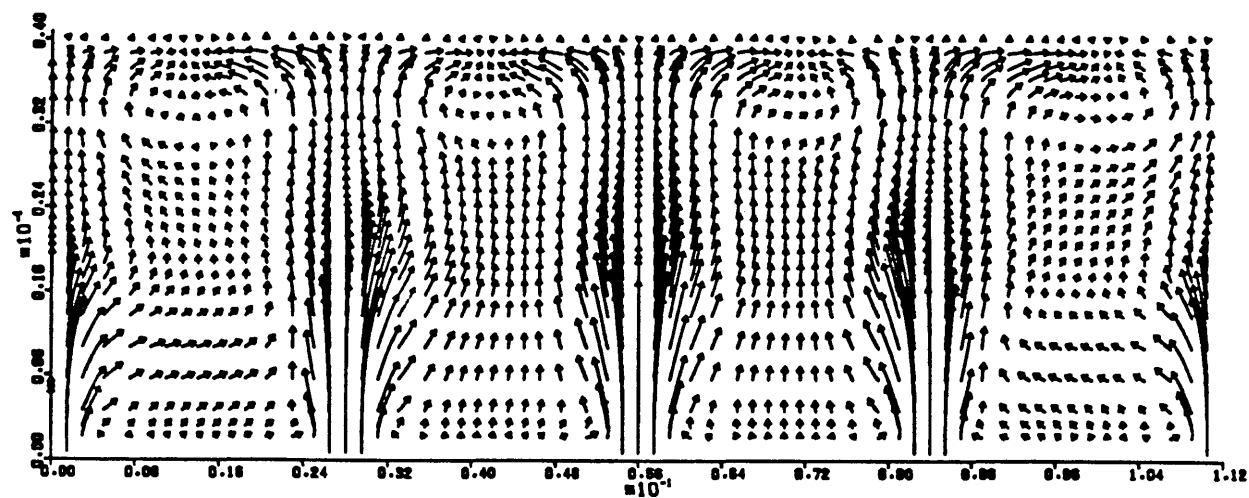


Figure 5.22 Streakline Plot for Increased Mass Flow at Slot 3 in Isothermal Flow

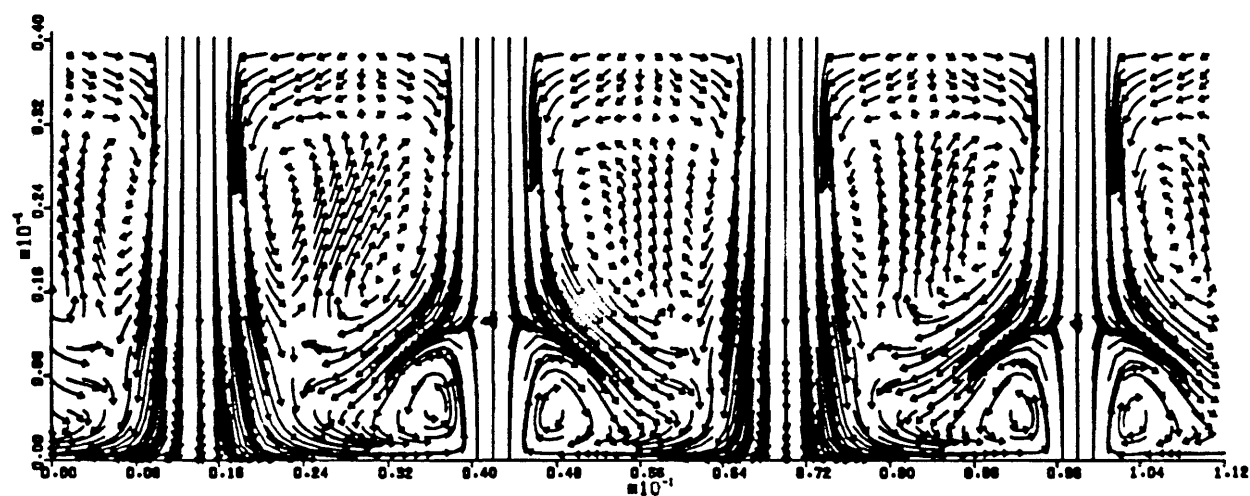
(a) $z=28\text{mm}$ (b) $z=0\text{mm}$

(c) $x=45\text{mm}$ (d) $x=74\text{mm}$ (e) $x=100\text{mm}$

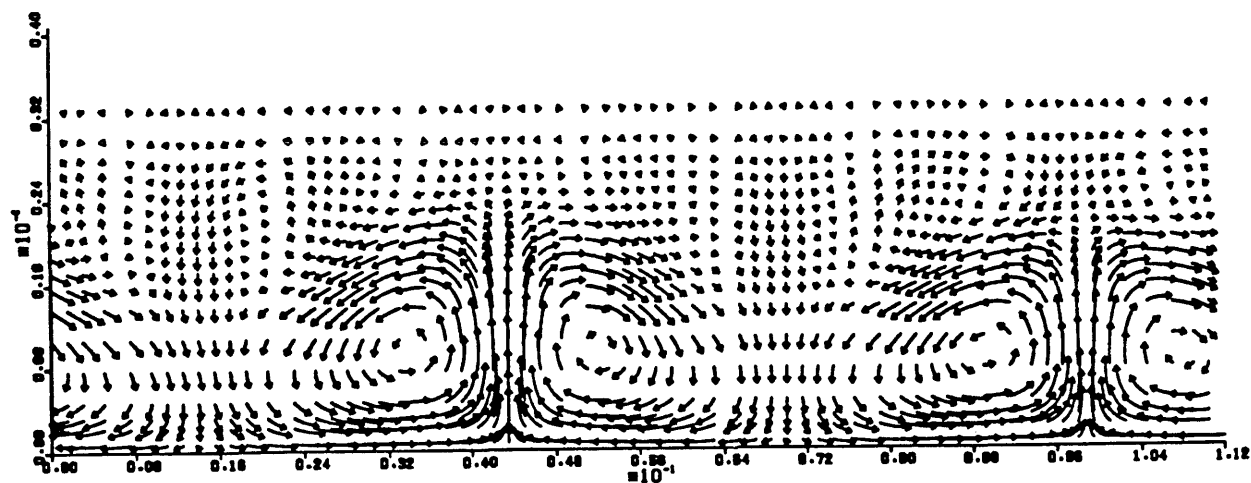
(c)



(d)



(e)



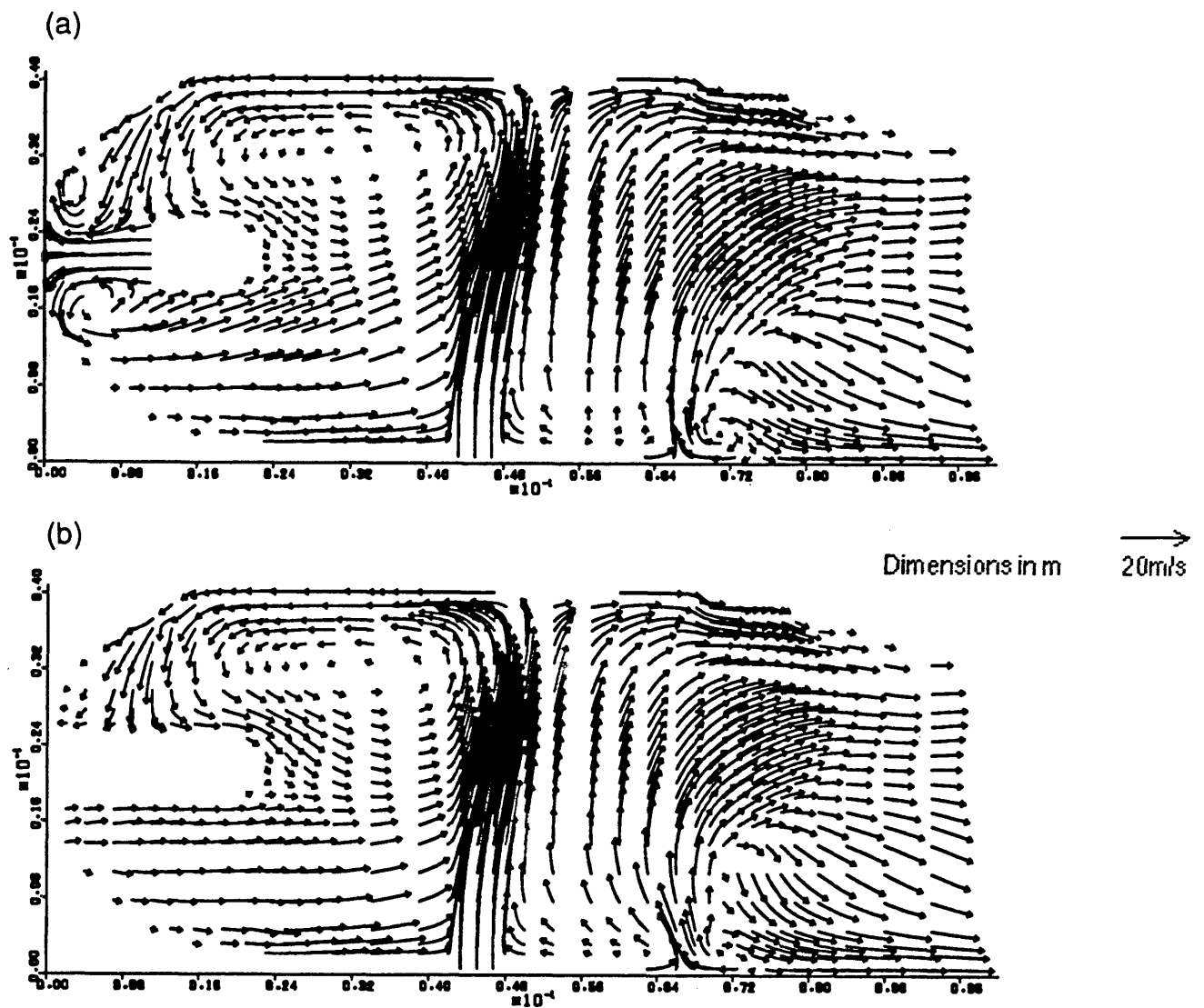


Figure 5.23 Streakline Plot for Increased Mass Flow at Slot 4 in Isothermal Flow

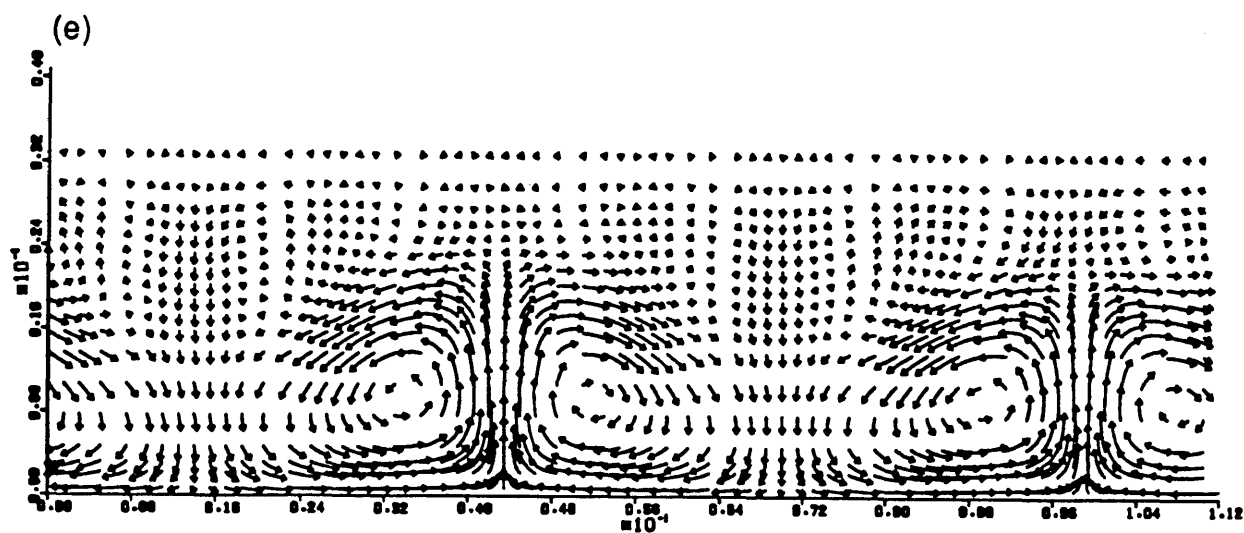
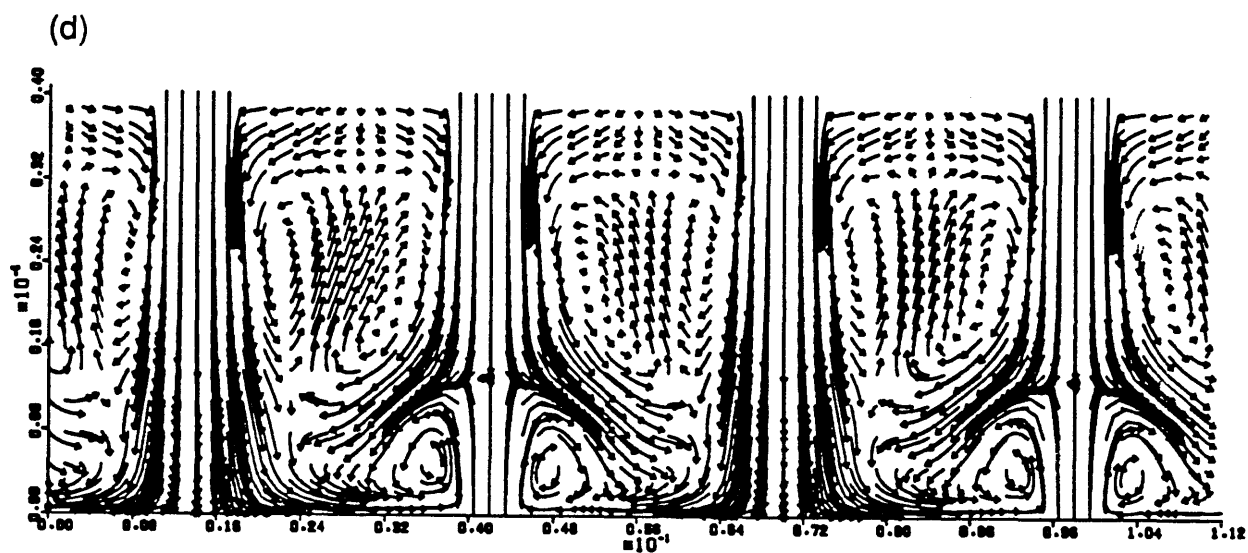
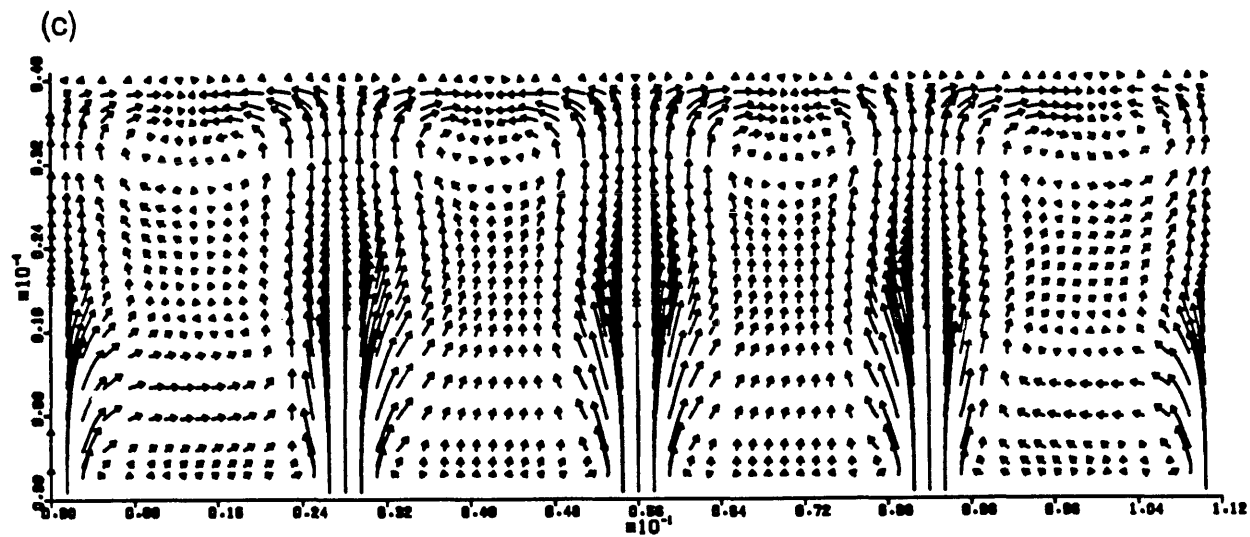
(a) $z=28\text{mm}$

(b) $z=0\text{mm}$

(c) $x=45\text{mm}$

(d) $x=74\text{mm}$

(e) $x=100\text{mm}$



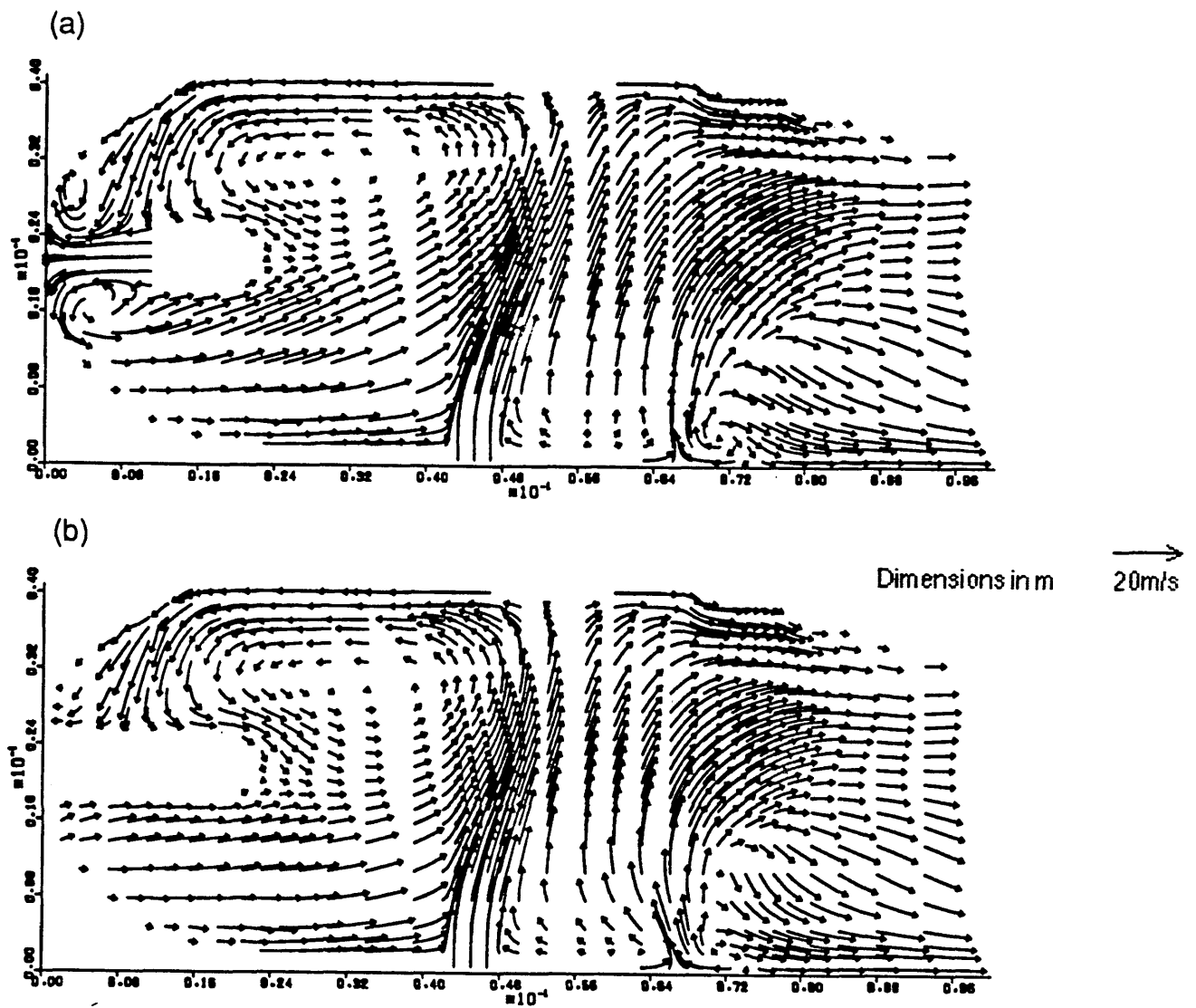


Figure 5.24 Streakline Plot for Increased Mass Flow at Slot 3 and 4 in Isothermal

Flow

(a) $z=28\text{mm}$

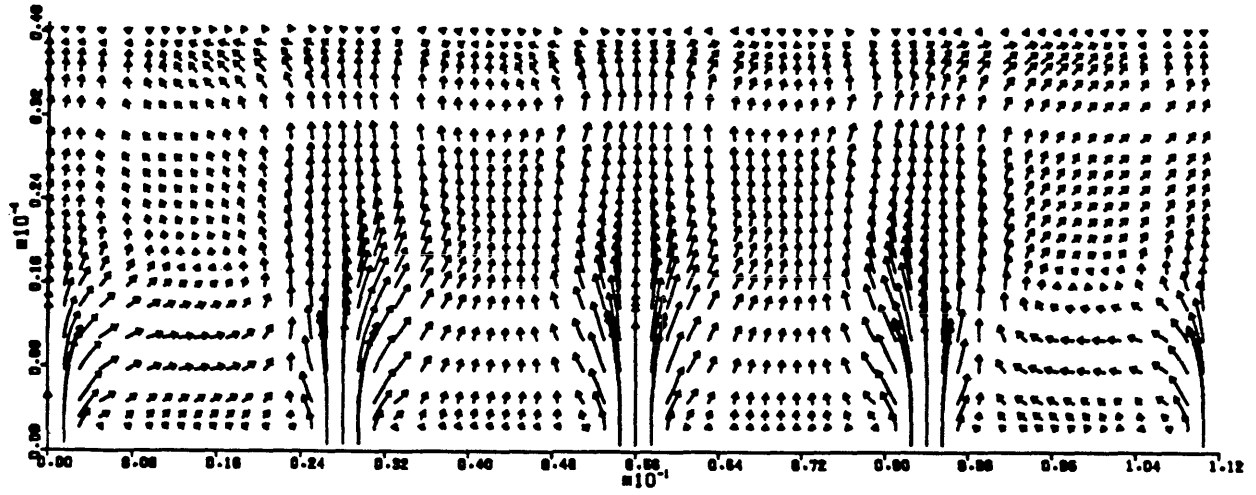
(b) $z=0\text{mm}$

(c) $x=45\text{mm}$

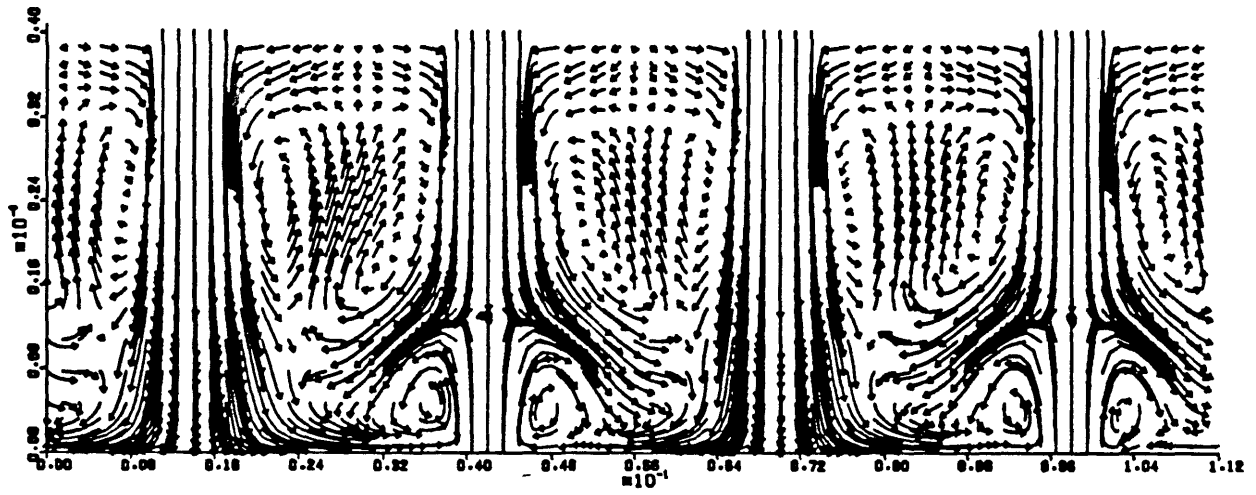
(d) $x=74\text{mm}$

(e) $x=100\text{mm}$

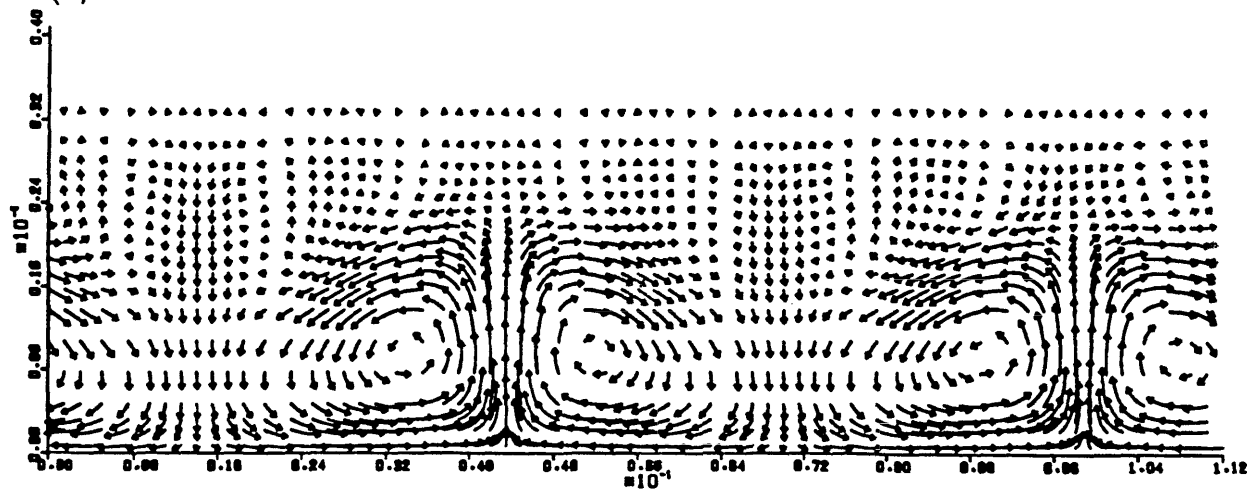
(c)



(d)



(e)



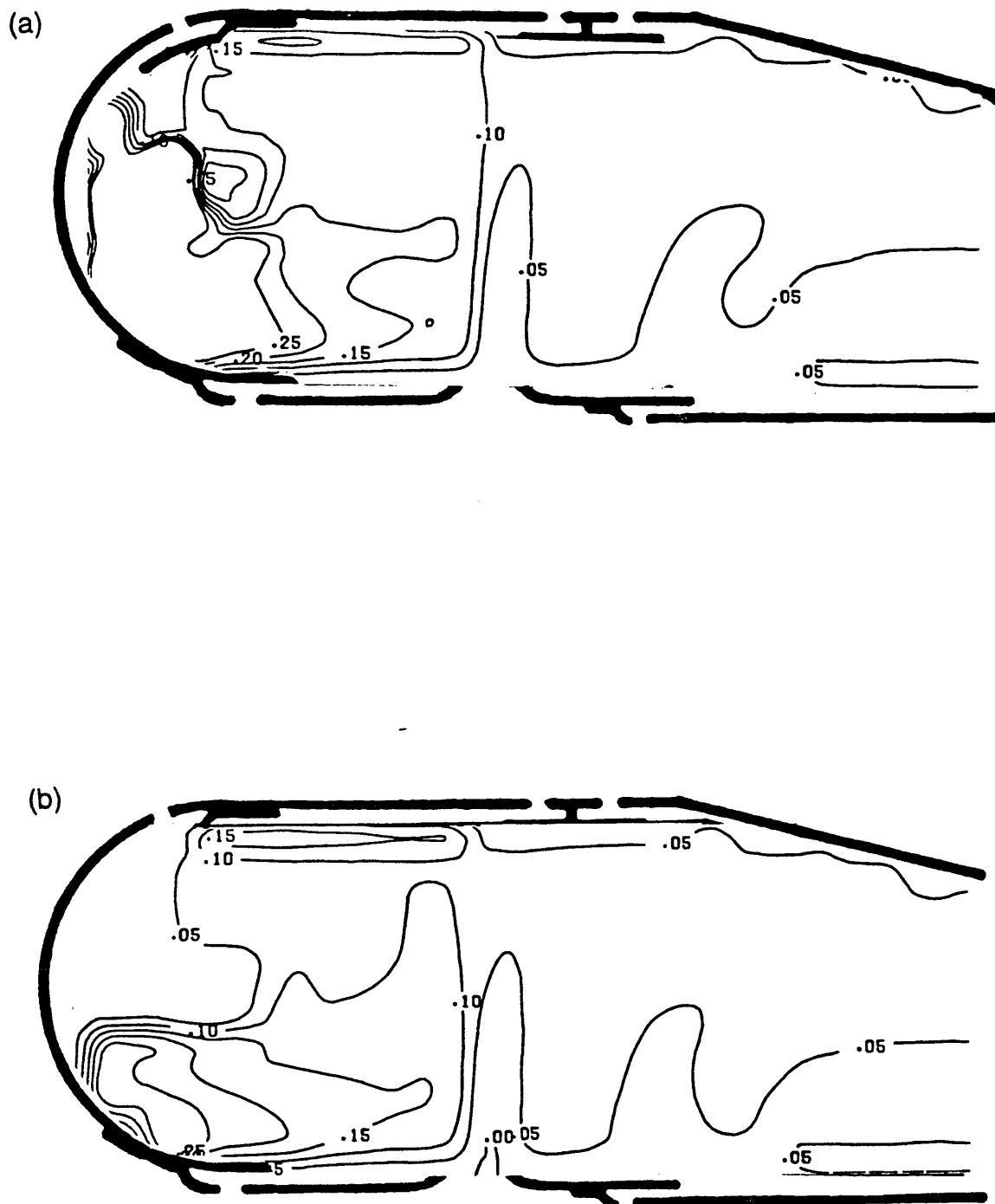


Figure 5.25 Predicted Passive Scalar Distributions for Non-Uniform Velocity
Profile at Vaporiser Exit in Isothermal Flow
(a) $z=28\text{mm}$ (b) $z=0\text{mm}$

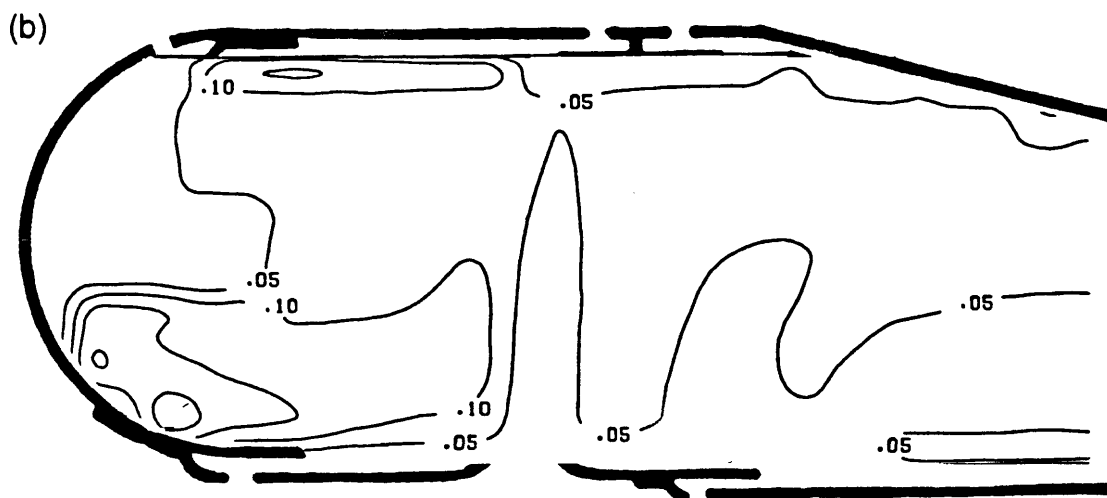
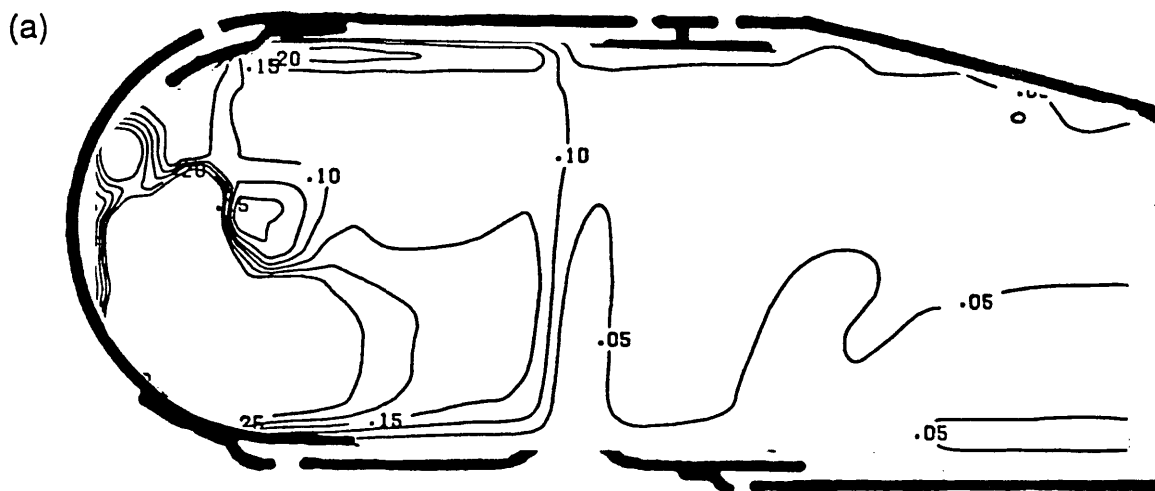


Figure 5.26 Predicted Passive Scalar Distributions for Second Component
 Velocity of 15% in Isothermal Flow
 (a) $z=28\text{mm}$ (b) $z=0\text{mm}$

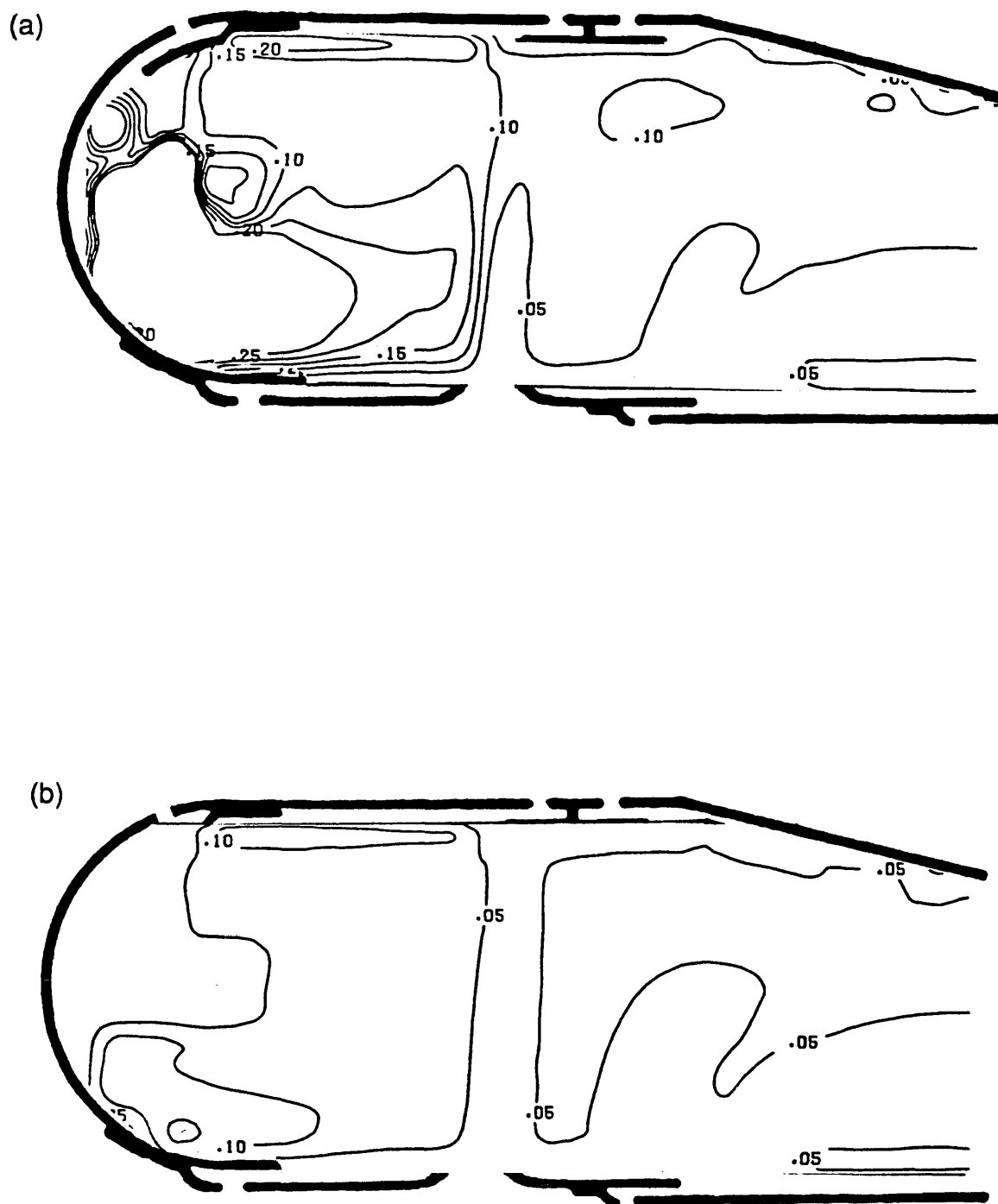


Figure 5.27 Predicted Passive Scalar Distributions for Second Component
Velocity of 25% in Isothermal Flow
(a) $z=28\text{mm}$ (b) $z=0\text{mm}$

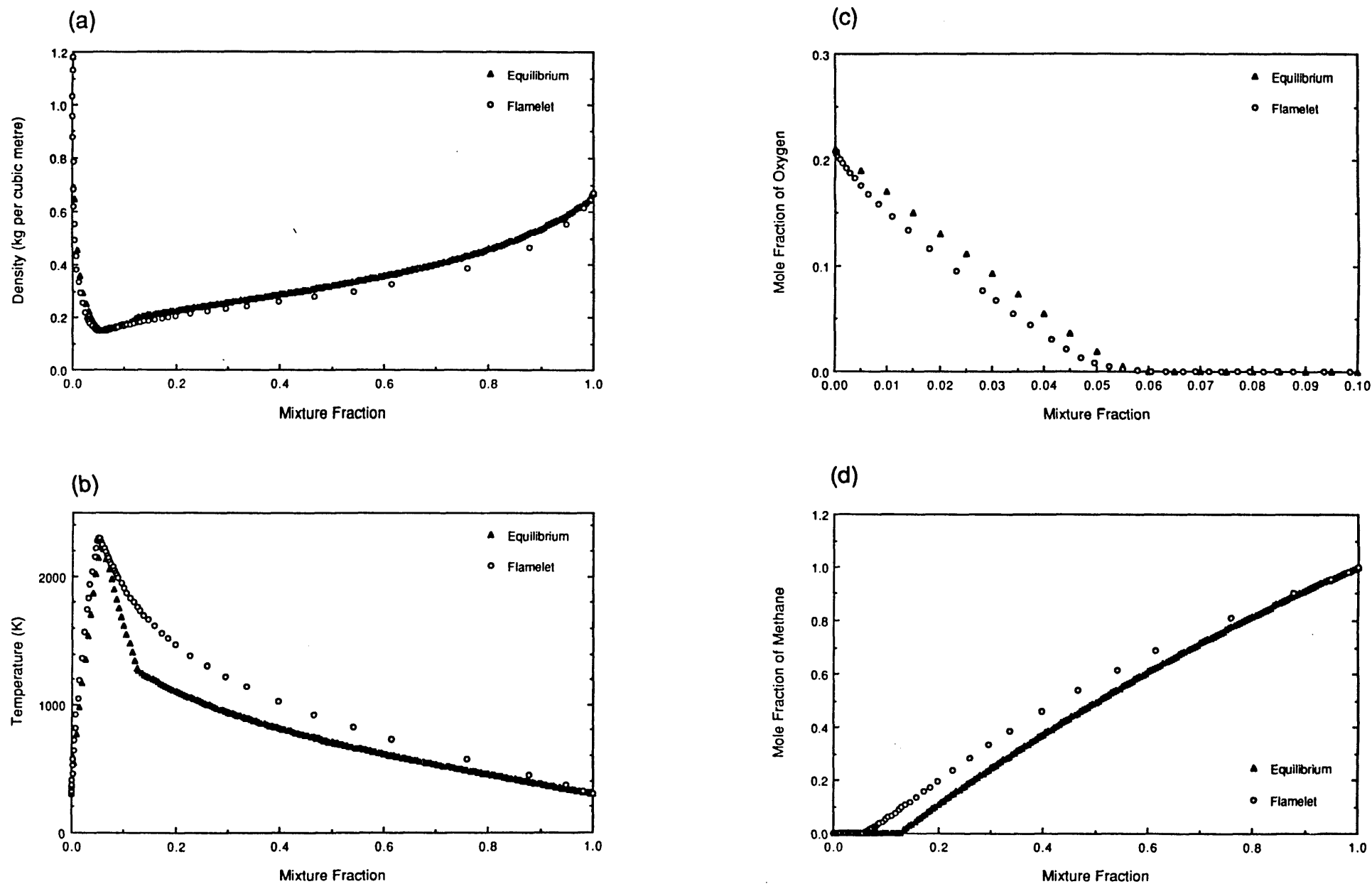


Figure 5.28 Distributions of Thermochemical Properties for the Equilibrium and Laminar Flamelet Models

(a) Density

(b) Temperature

(c) O₂

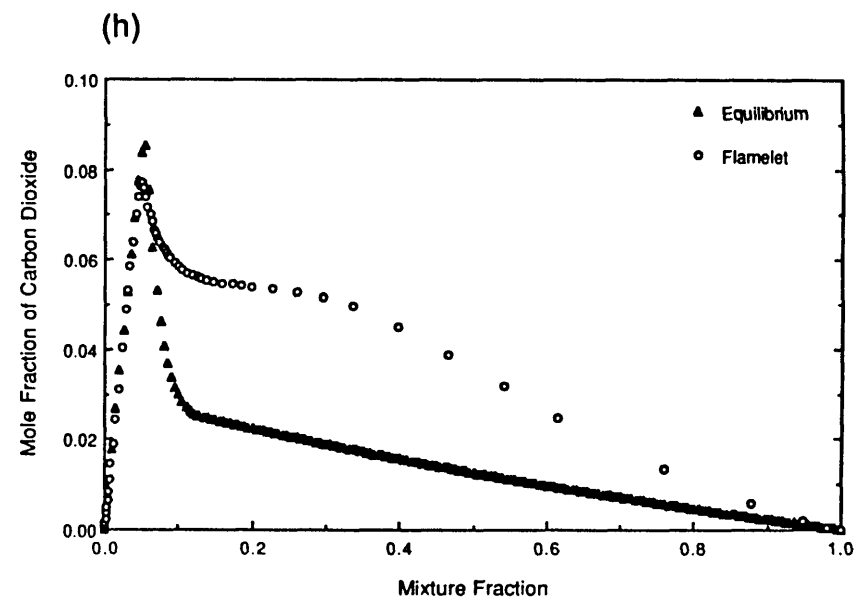
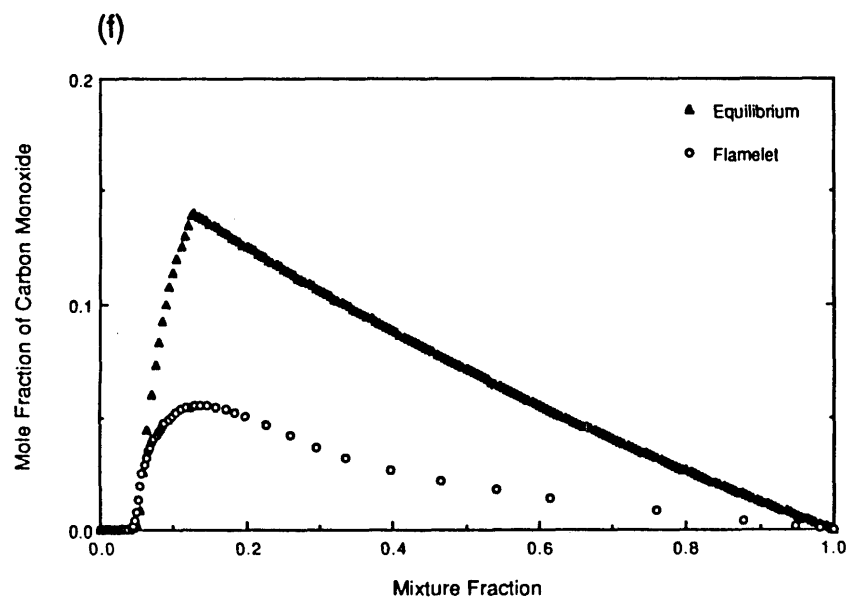
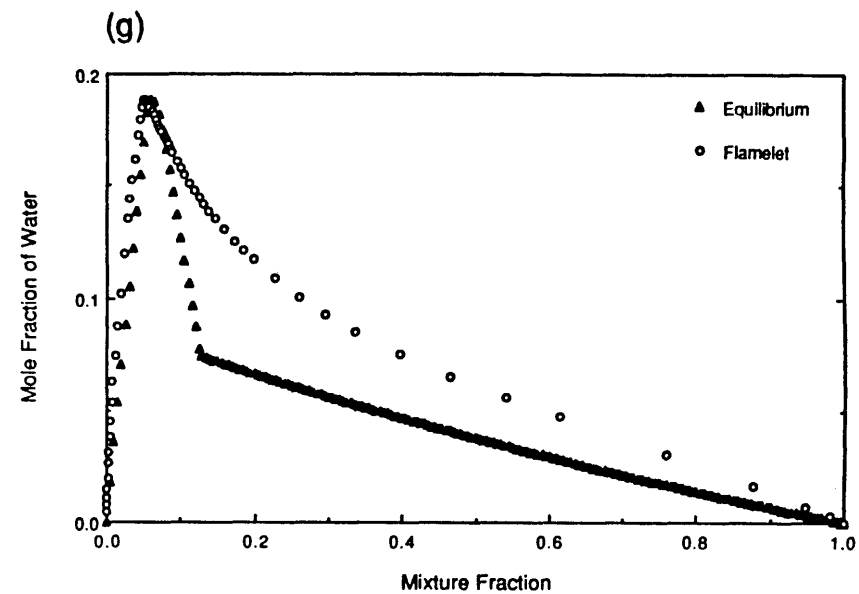
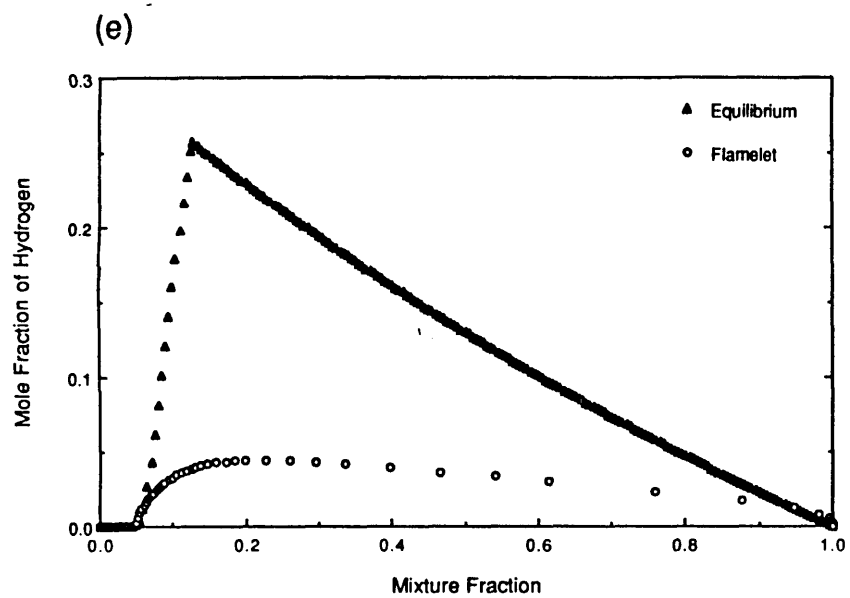
(d) CH₄

(e) H₂

(f) CO

(g) H₂O

(h) CO₂



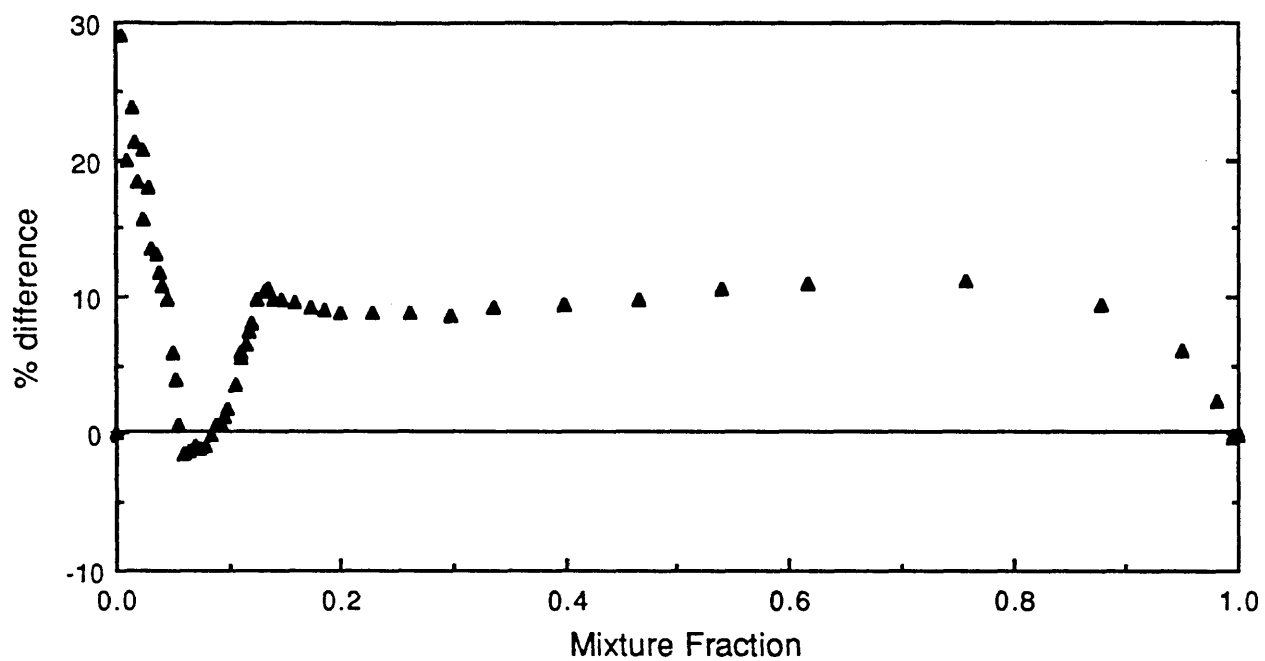


Figure 5.29 Percentage Difference in Density for the Equilibrium and Laminar Flamelet Models

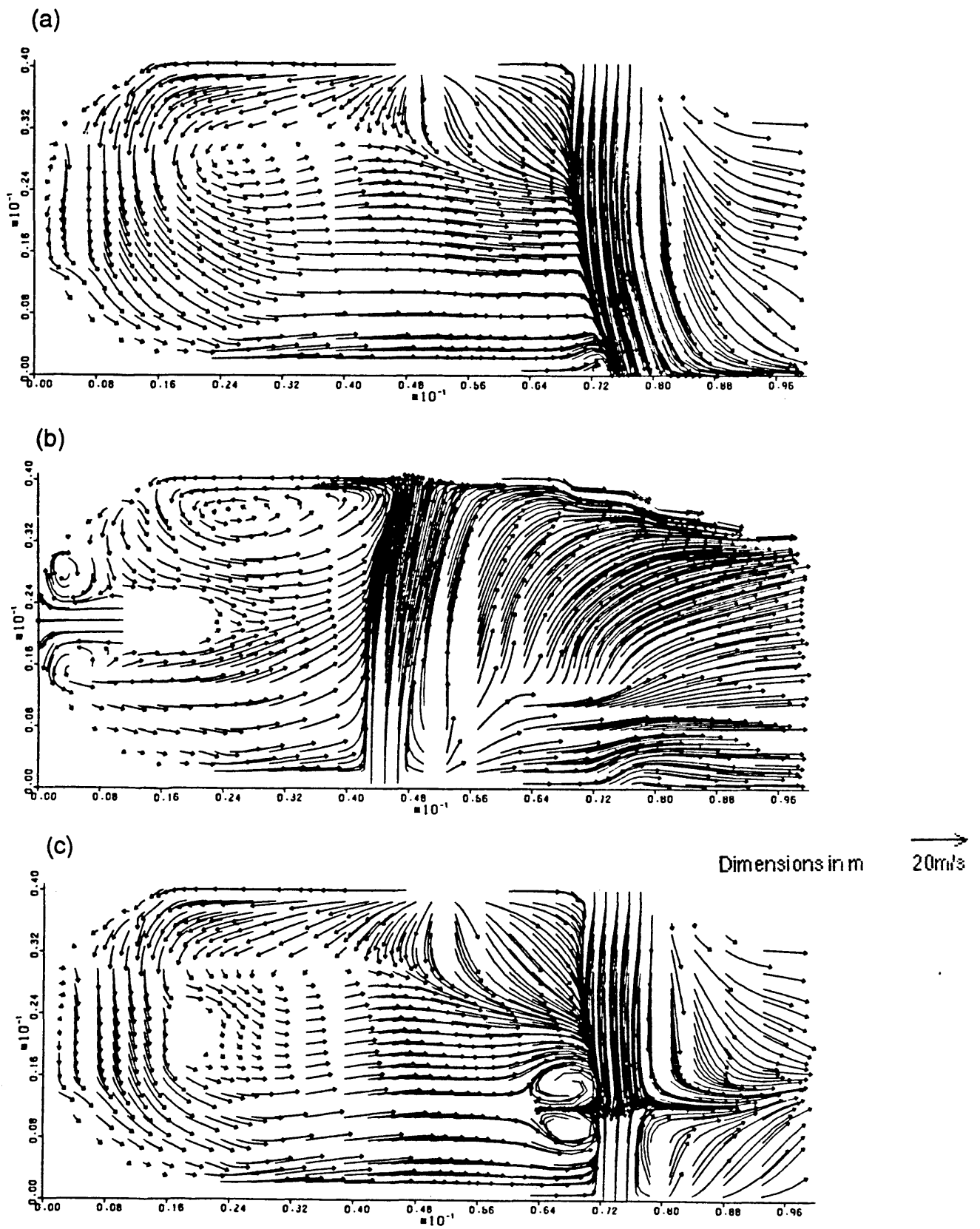


Figure 5.30 Streakline Plot of Combusting Flow of Combustor for $T_{inlet}=315K$ and

AFR=29

(a) $z=-42mm$

(b) $z=-28mm$

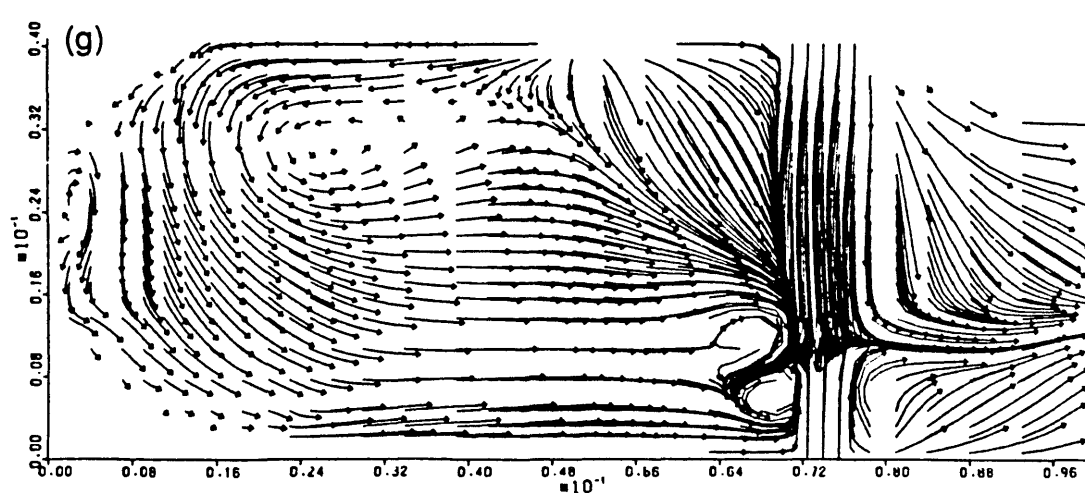
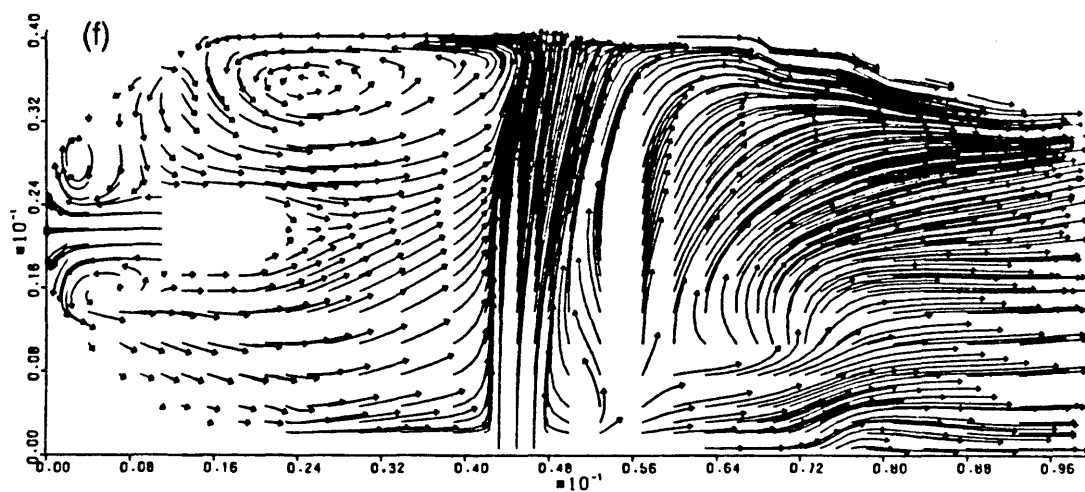
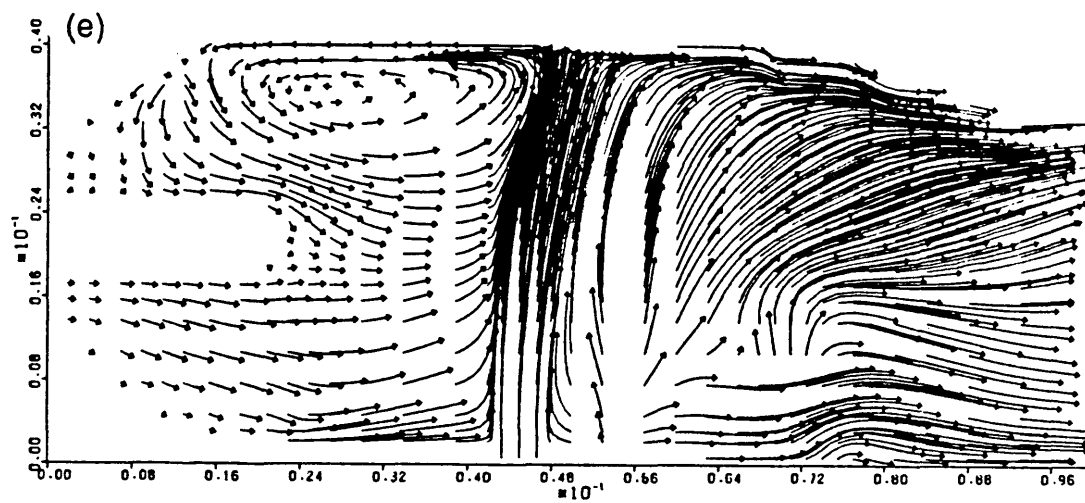
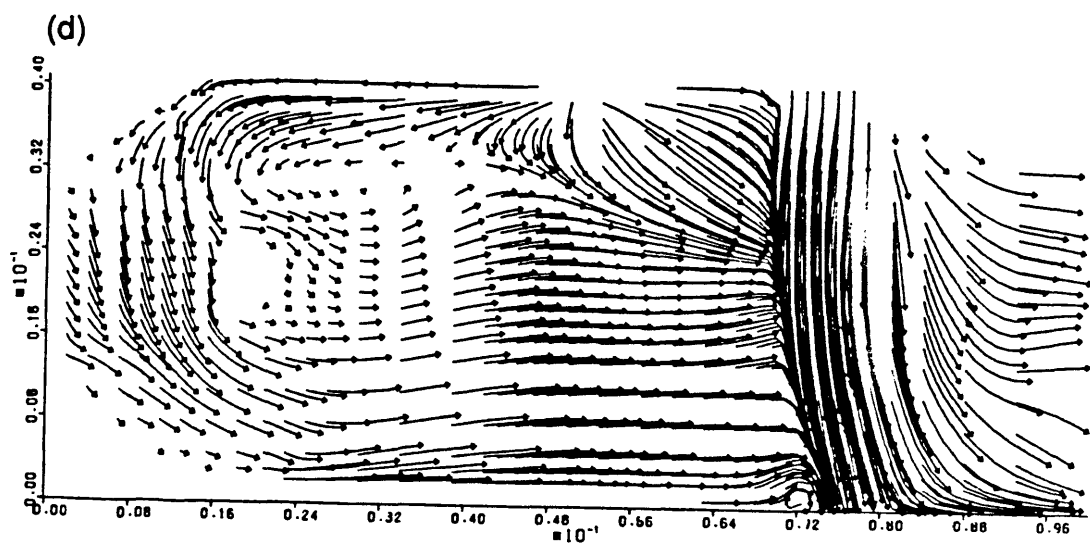
(c) $z=-14mm$

(d) $z=0mm$

(e) $z=14mm$

(f) $z=28mm$

(g) $z=42mm$



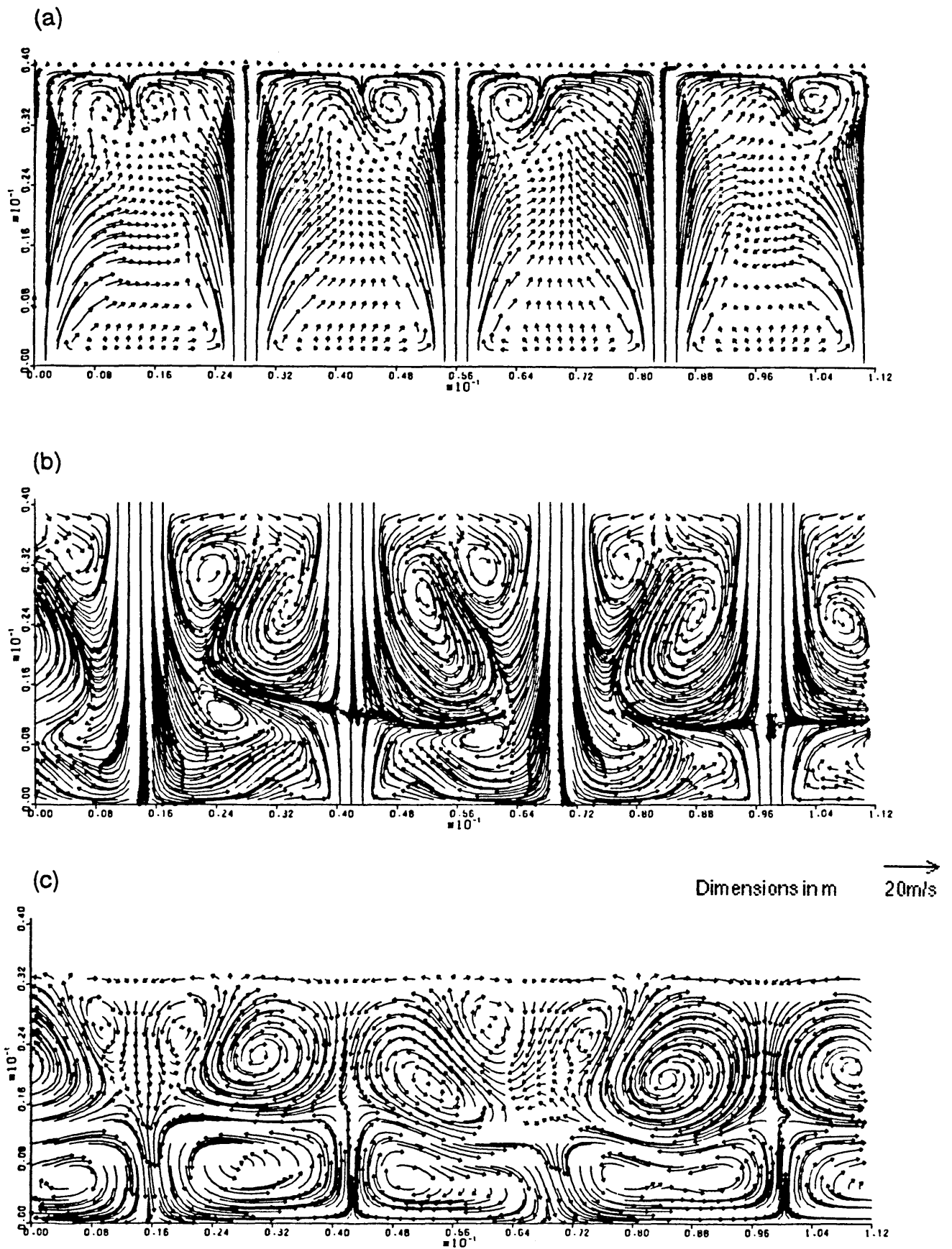


Figure 5.31 Streakline Plot of Combusting Flow of Combustor in Cross-Stream

Planes for $T_{\text{inlet}}=315\text{K}$ and $\text{AFR}=29$

(a) $x=45\text{mm}$

(b) $x=74\text{mm}$

(c) $x=100\text{mm}$

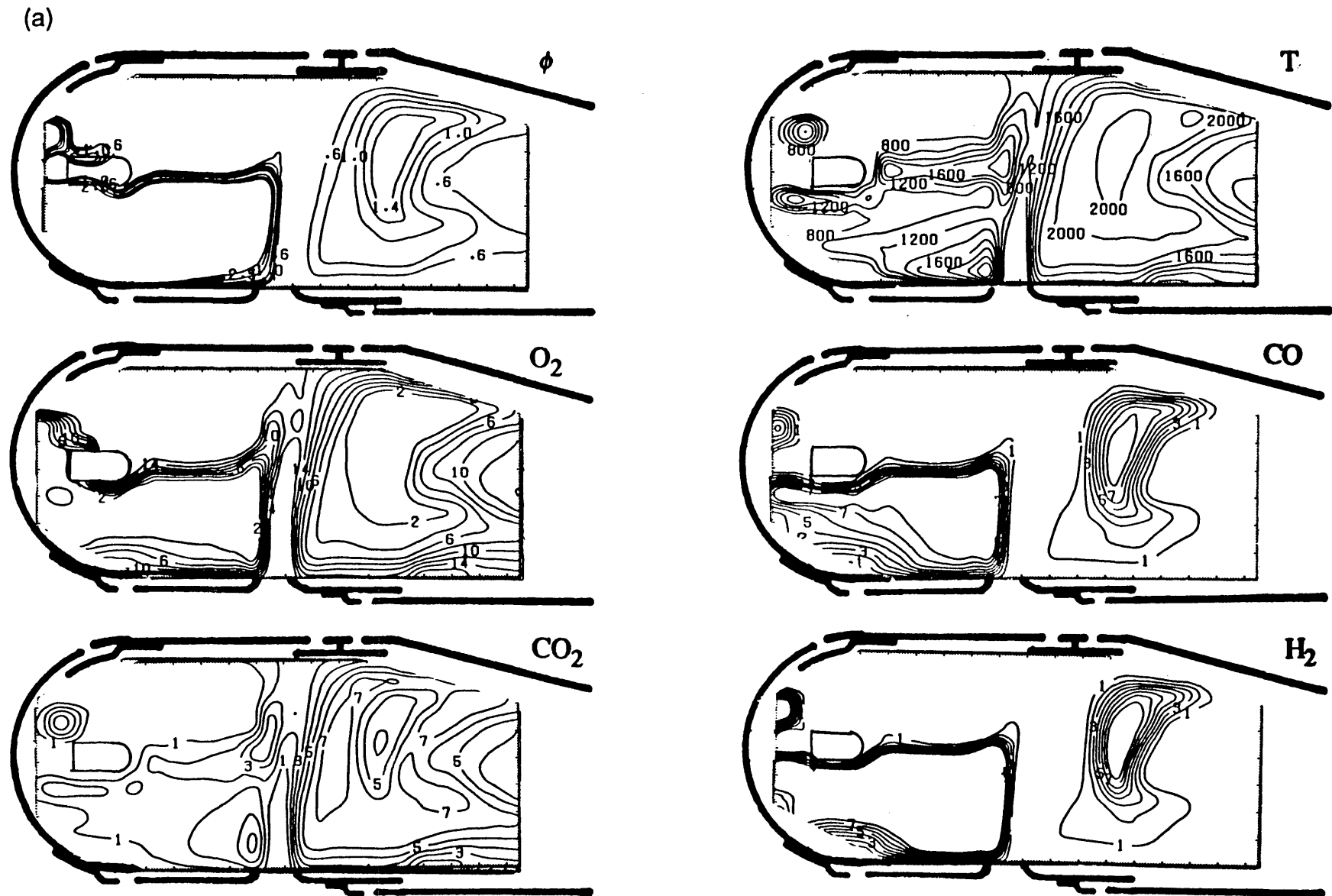


Figure 5.32 Predicted Species Concentrations Distributions with Equilibrium

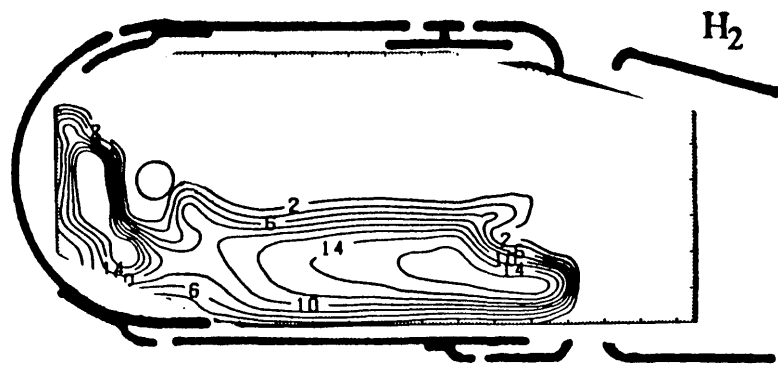
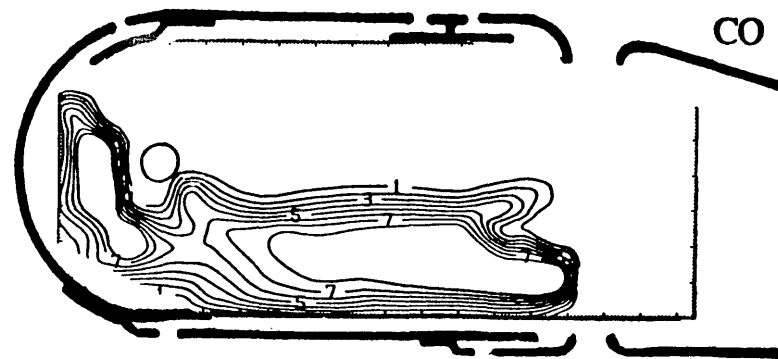
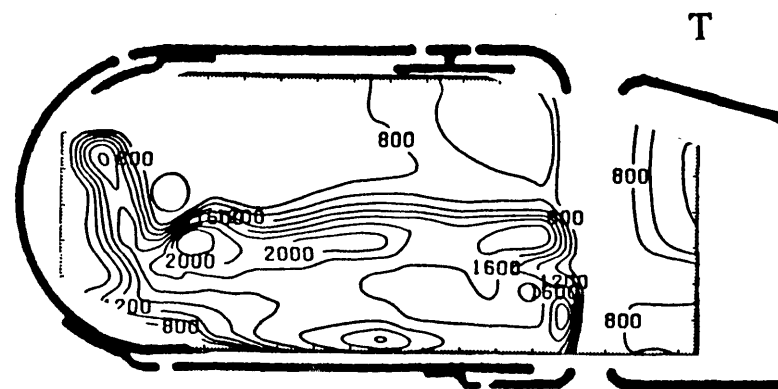
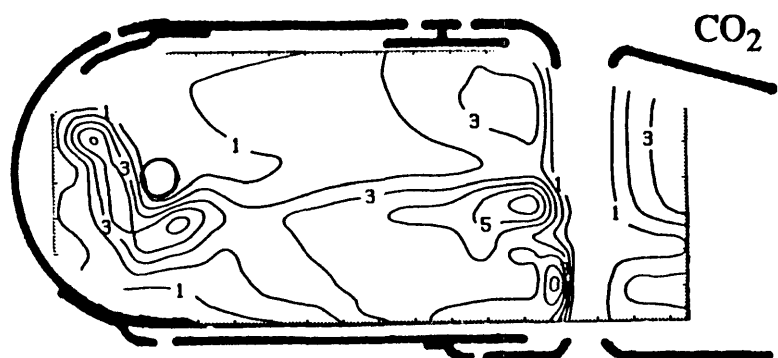
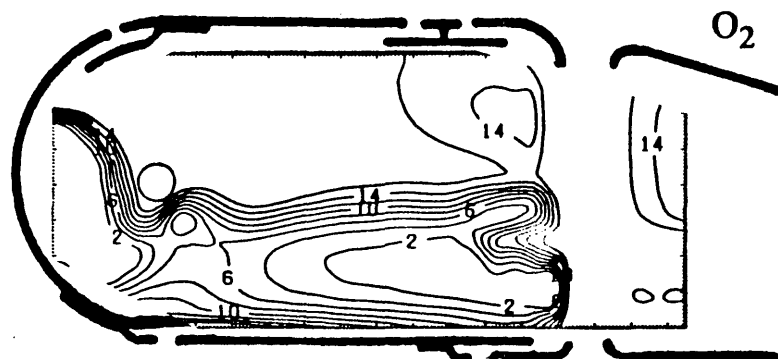
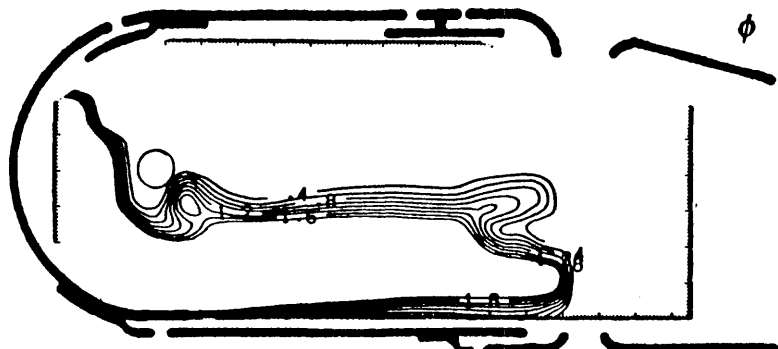
Model for $T_{inlet}=315K$ and $AFR=29$

(a) $z=-28mm$

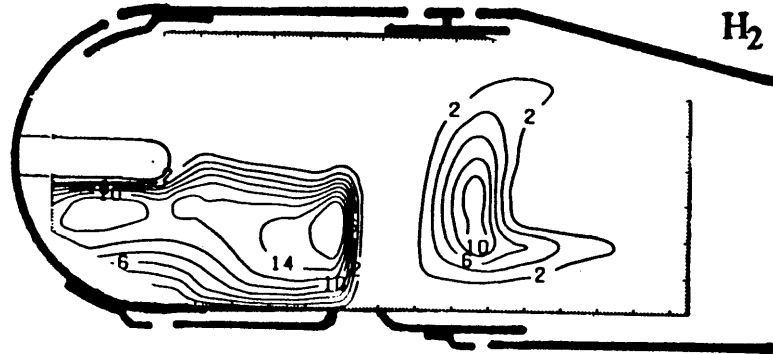
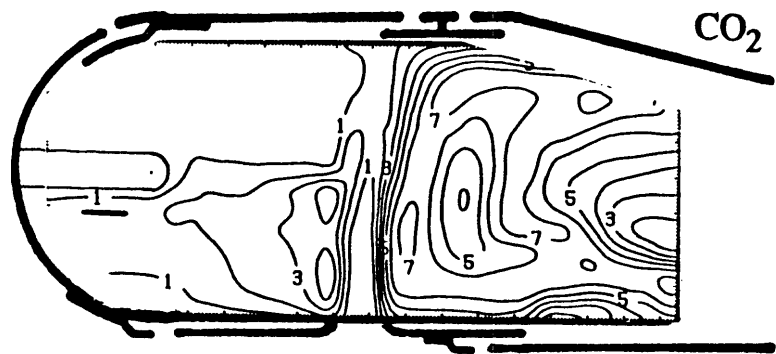
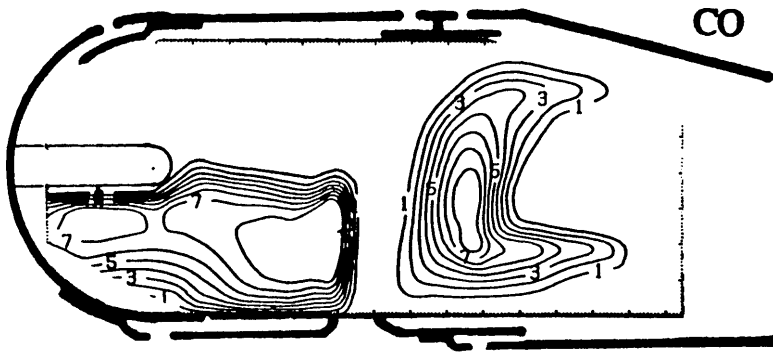
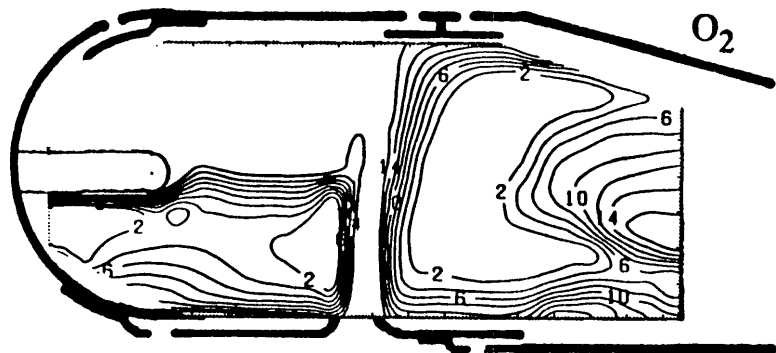
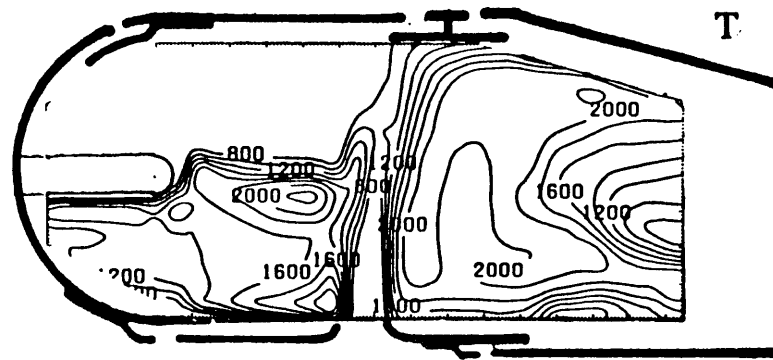
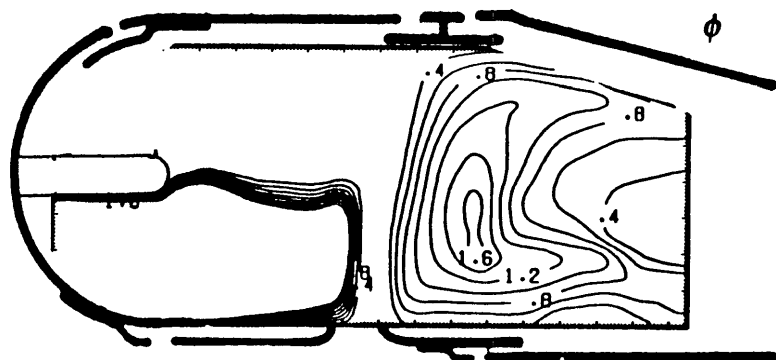
(b) $z=-14mm$

(c) $z=0mm$

(b)



(c)



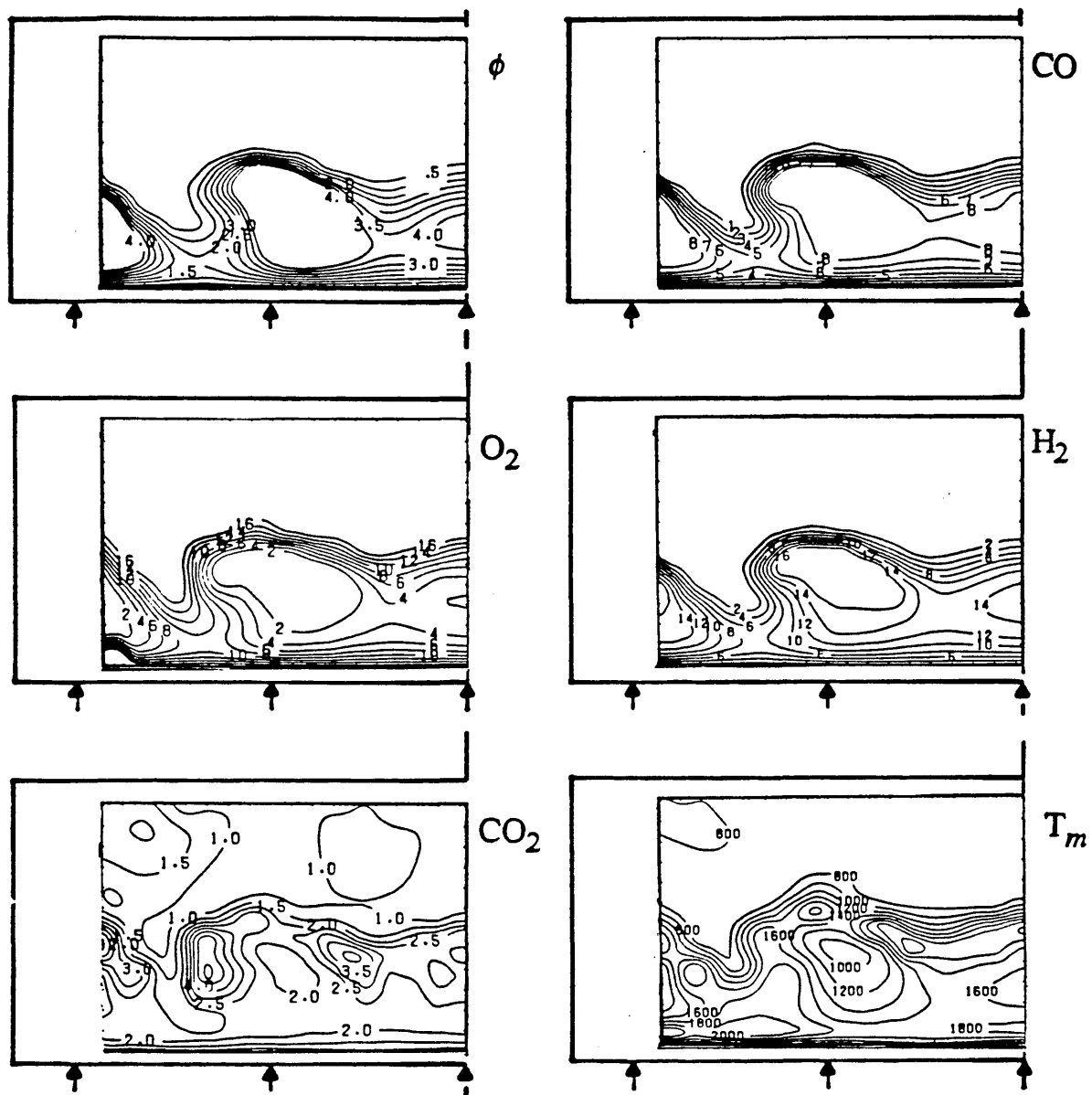


Figure 5.33 Predicted Species Concentrations Distributions with Equilibrium Model for $T_{inlet}=315K$ and $AFR=29$ at $x=36mm$

(a)

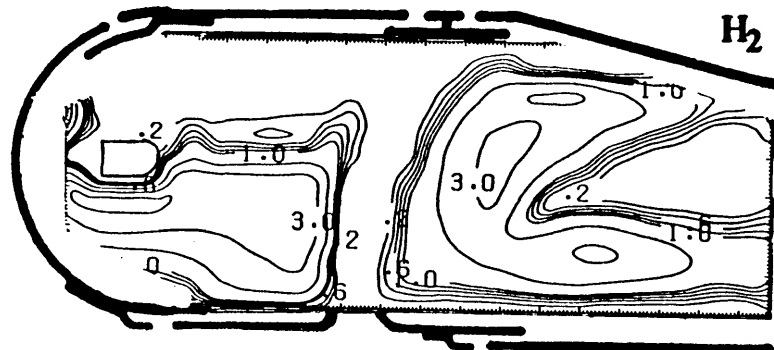
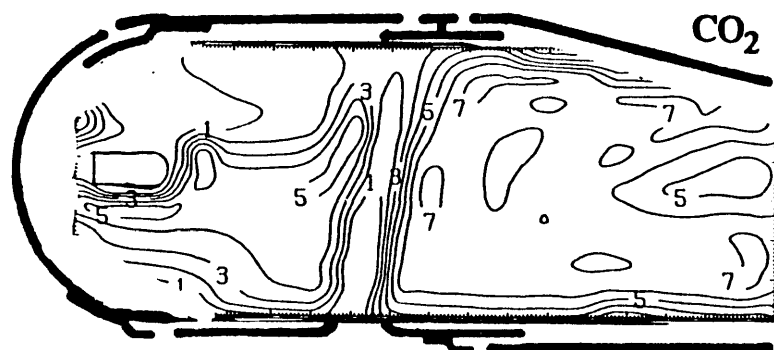
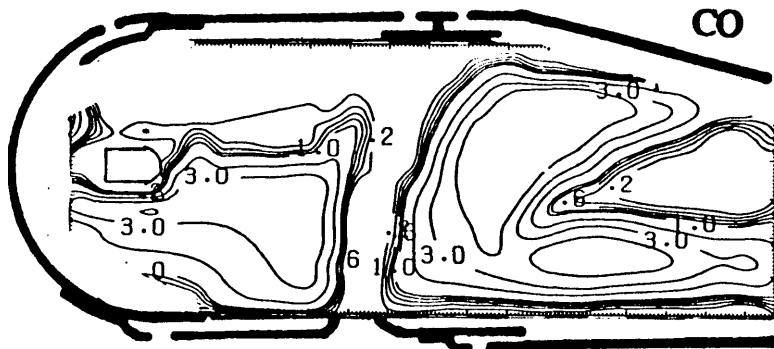
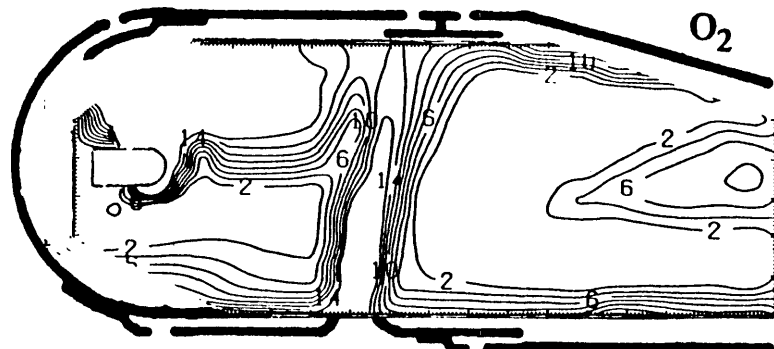
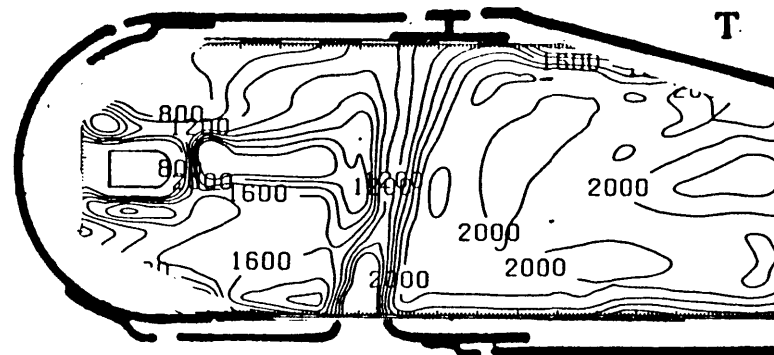
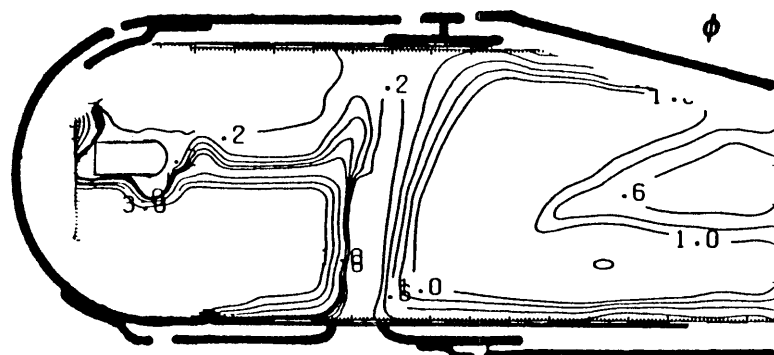


Figure 5.34 Predicted Species Concentrations Distributions with Laminar

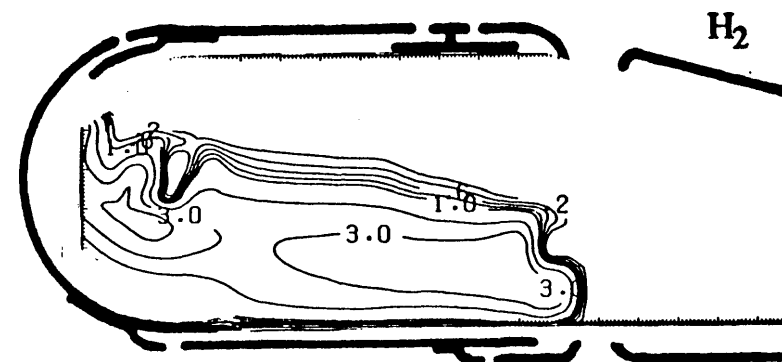
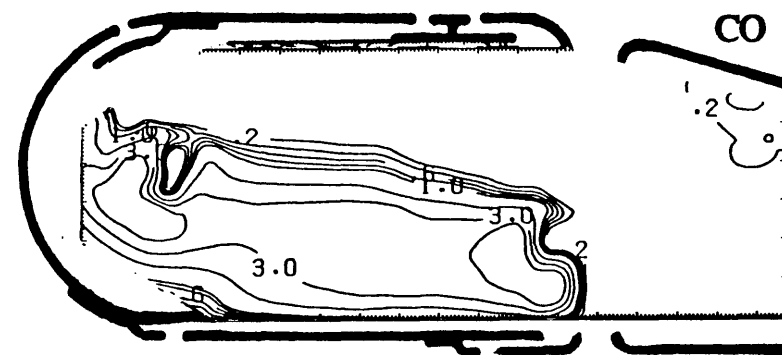
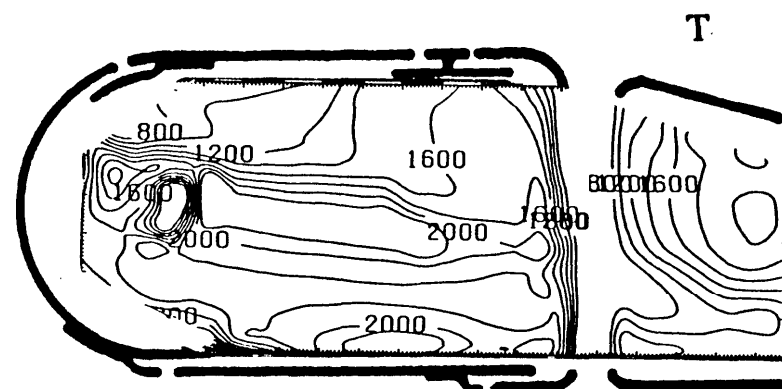
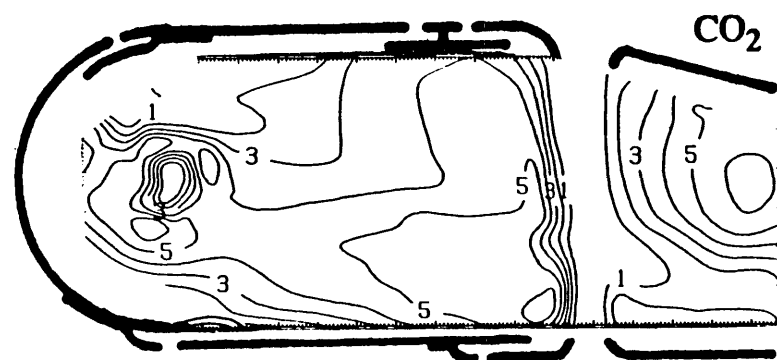
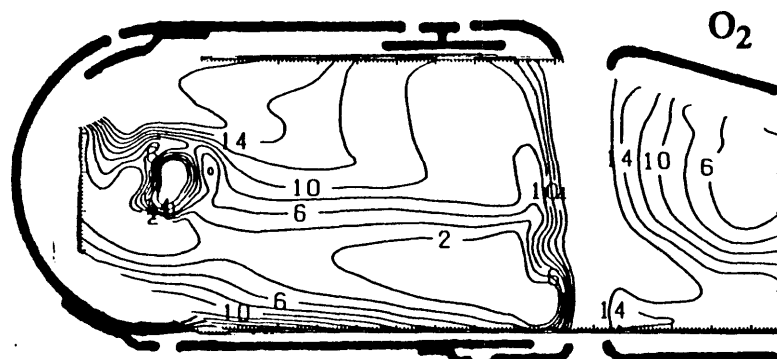
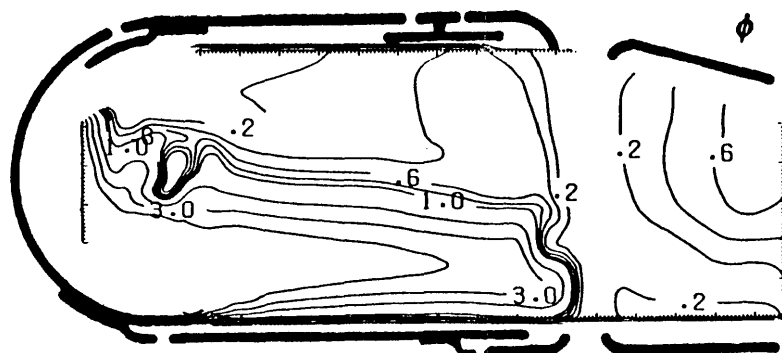
Flamelet Model for $T_{inlet}=315\text{K}$ and $AFR=29$

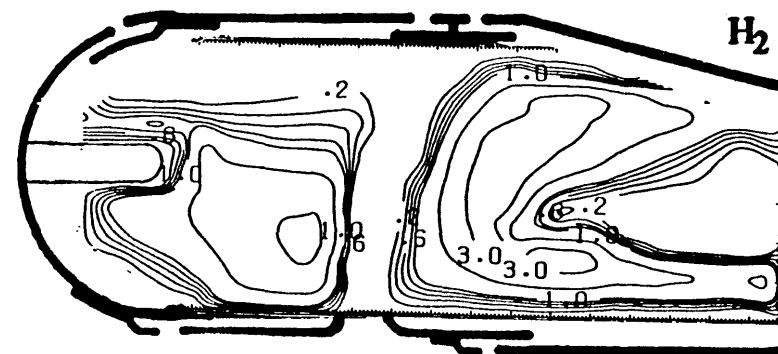
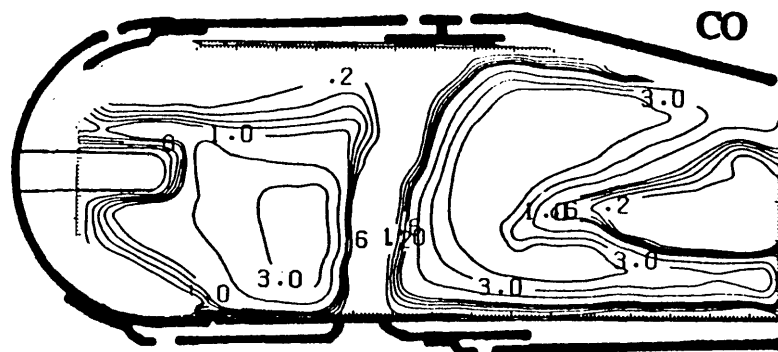
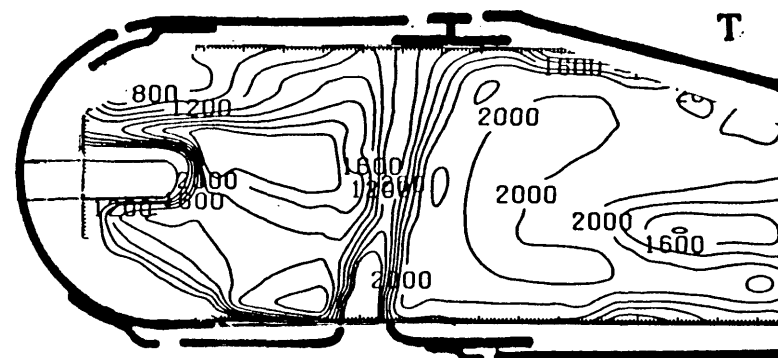
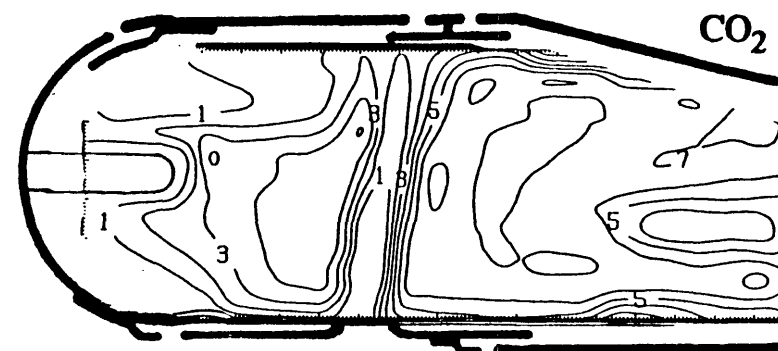
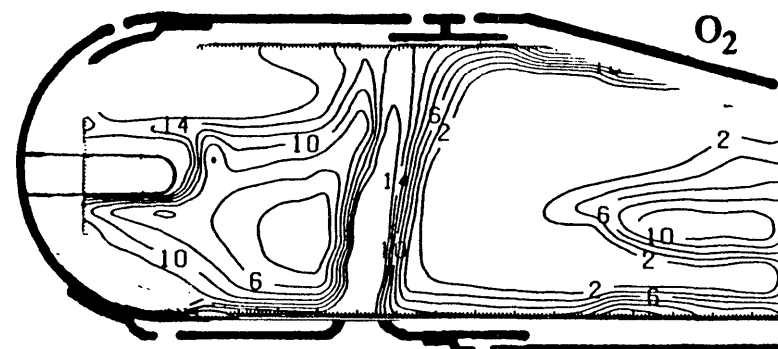
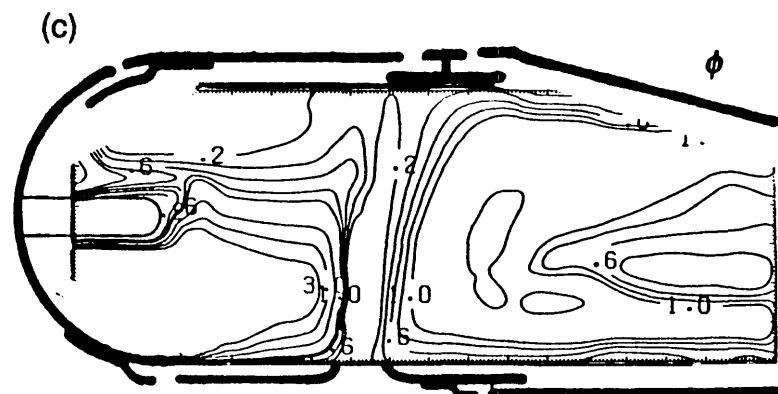
(a) $z=-28\text{mm}$

(b) $z=-14\text{mm}$

(c) $z=0\text{mm}$

(b)





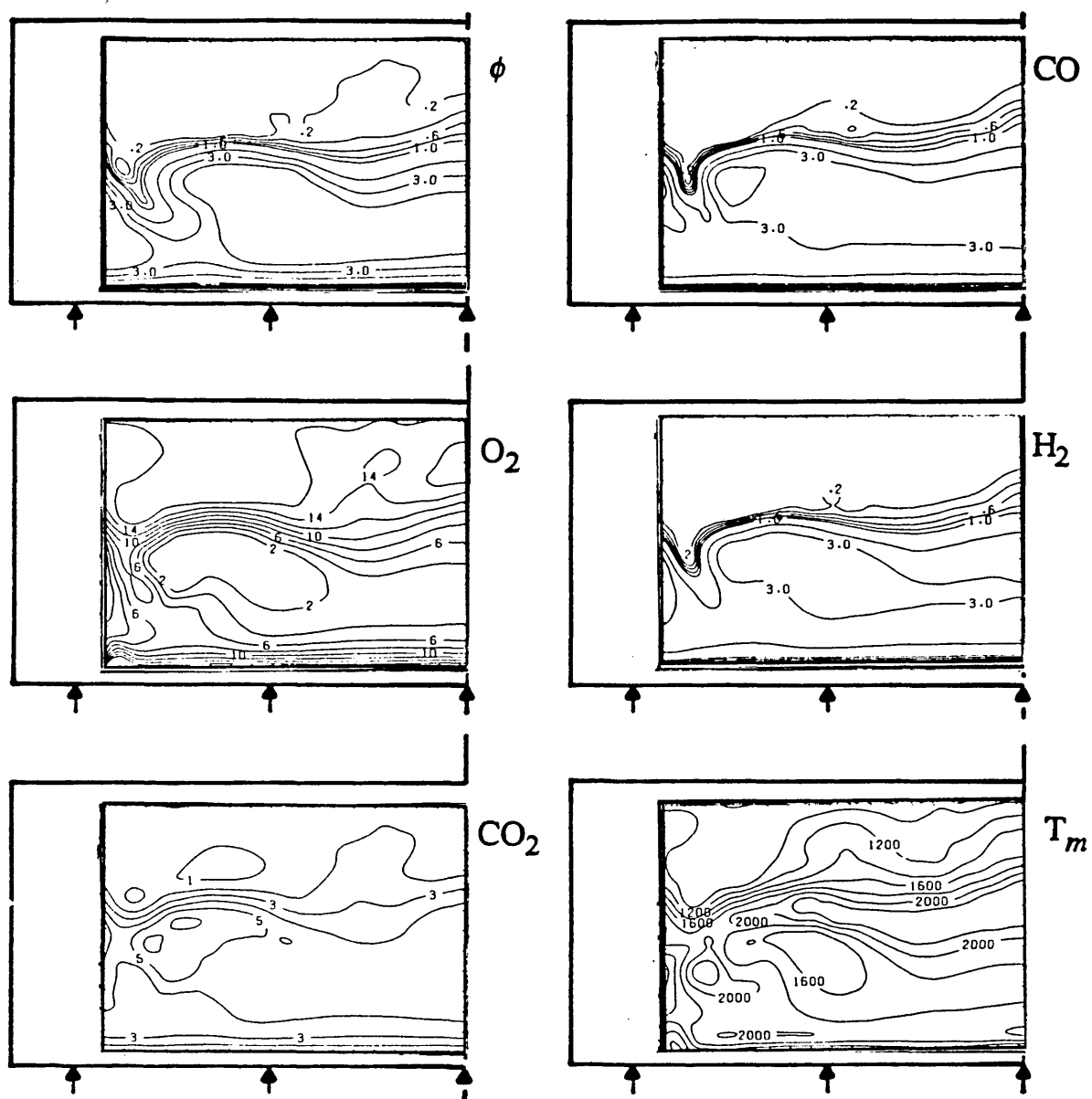


Figure 5.35 Predicted Species Concentrations Distributions with Laminar Flamelet Model for $T_{inlet}=315K$ and $AFR=29$ at $x=36mm$

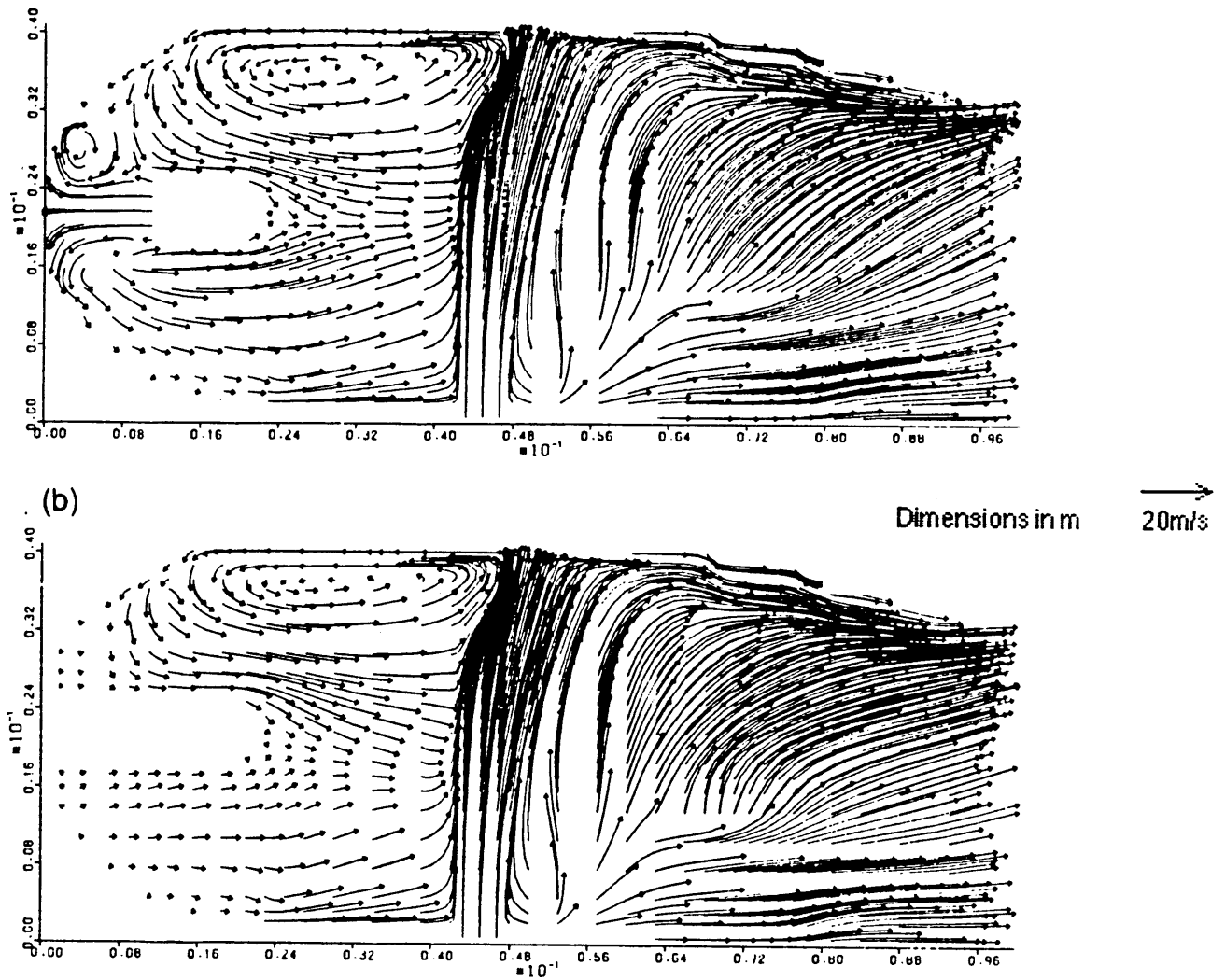


Figure 5.36 Streakline Plot of Combusting Flow of Combustor with Laminar

Flamelet Model for $T_{inlet}=315K$ and $AFR=29$

(a) $z=28mm$

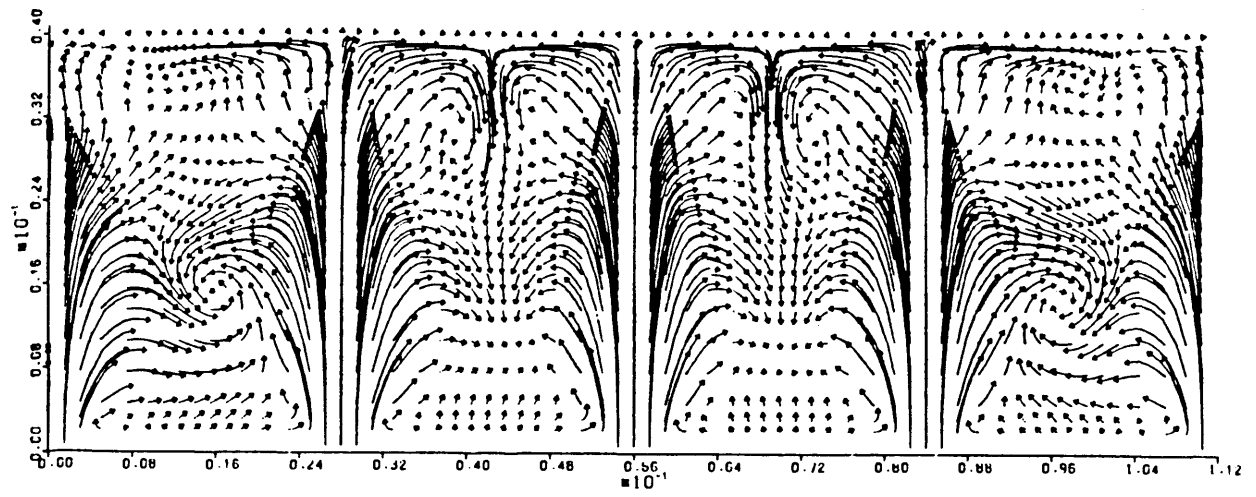
(b) $z=0mm$

(c) $x=45mm$

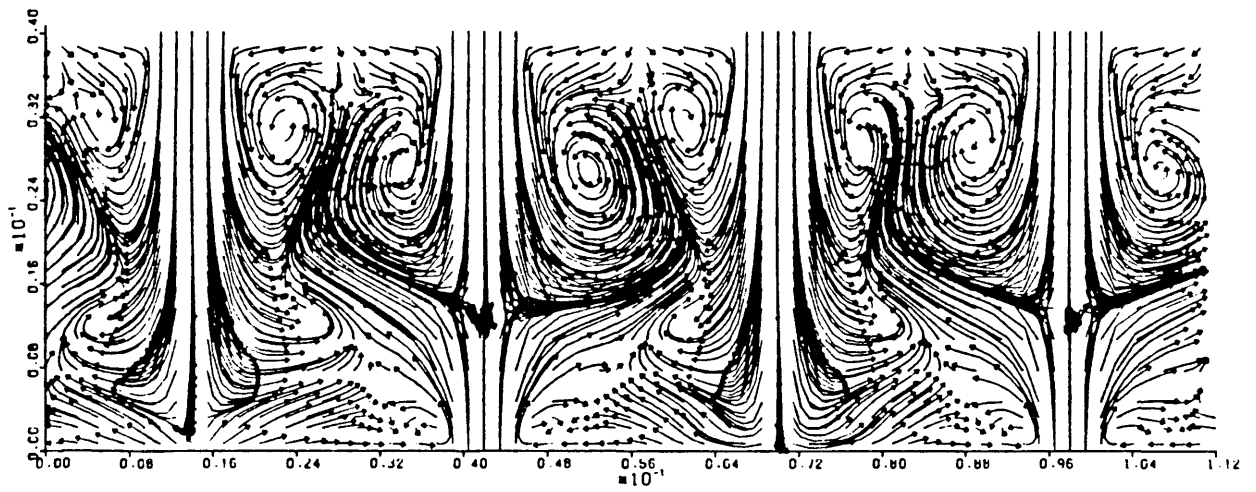
(d) $x=74mm$

(e) $x=100mm$

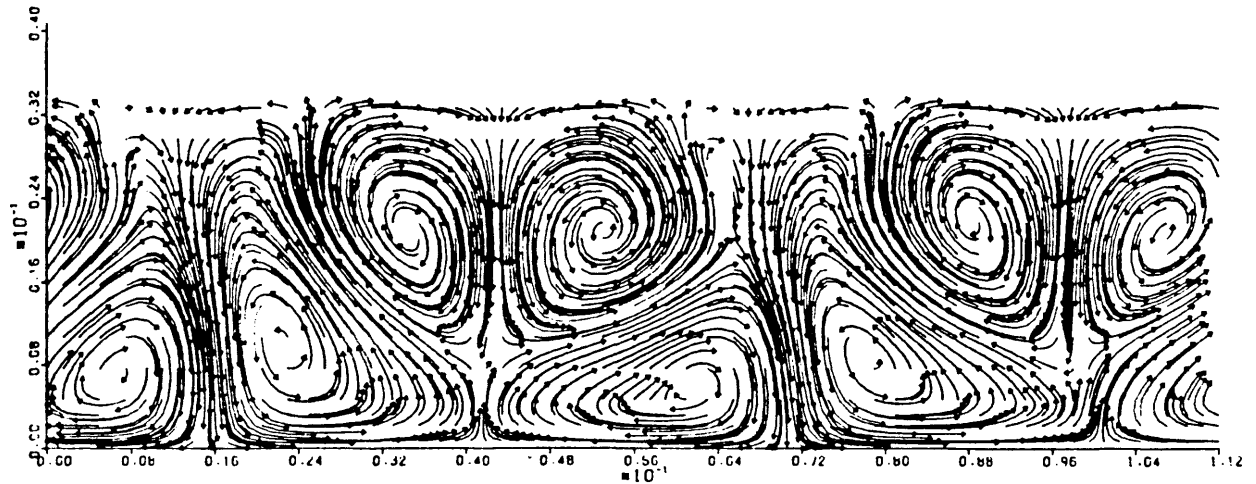
(c)



(d)



(e)



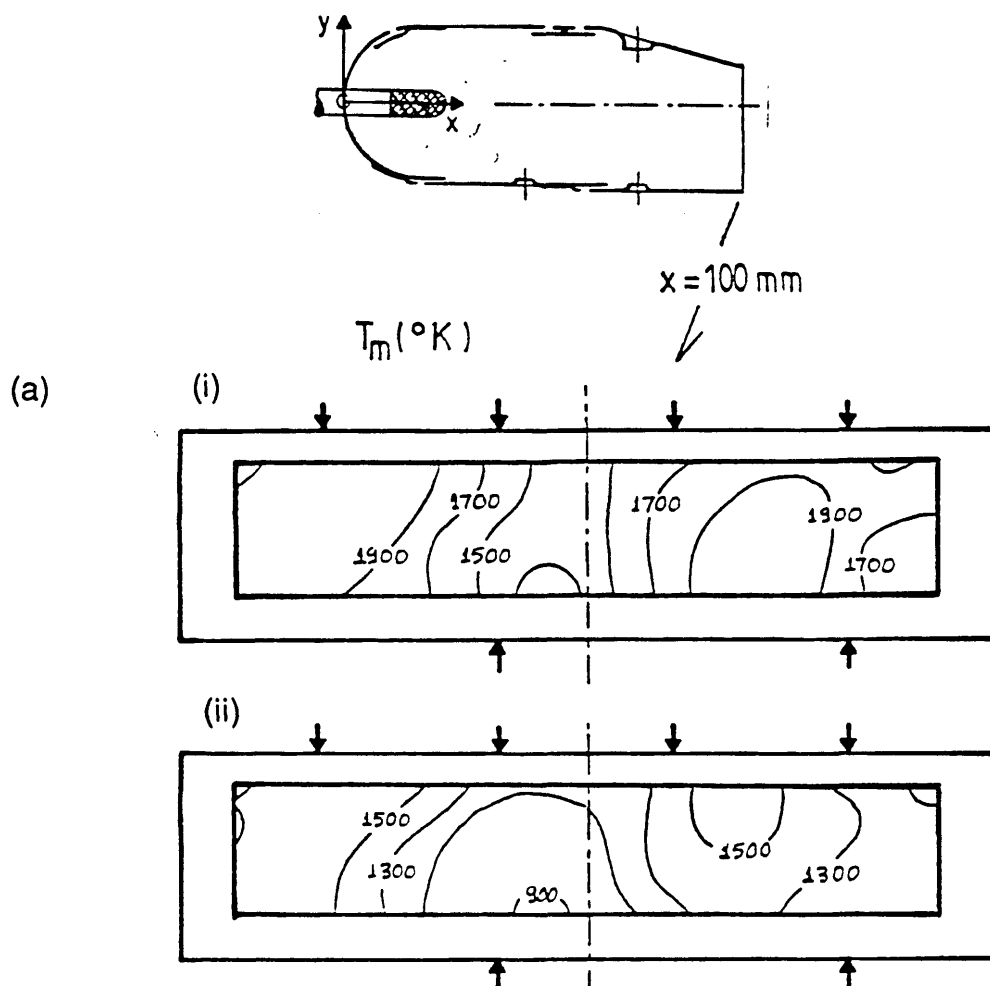


Figure 5.37 Measured Temperature and Species Concentrations with Natural

Gas as Fuel (Taken from Tse (1988))

(a) Mean Temperature Distributions

(i) $T_{inlet}=515K$ and $AFR=29$ in Exit Plane

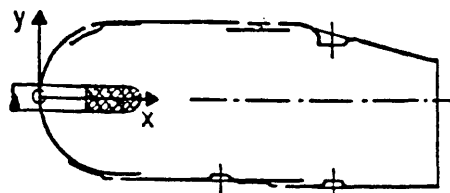
(ii) $T_{inlet}=515K$ and $AFR=50$ in Exit Plane

(b) Species Concentrations Distributions

(i) $T_{inlet}=515K$ and $AFR=29$ in Exit Plane

(ii) $T_{inlet}=515K$ and $AFR=50$ in Exit Plane

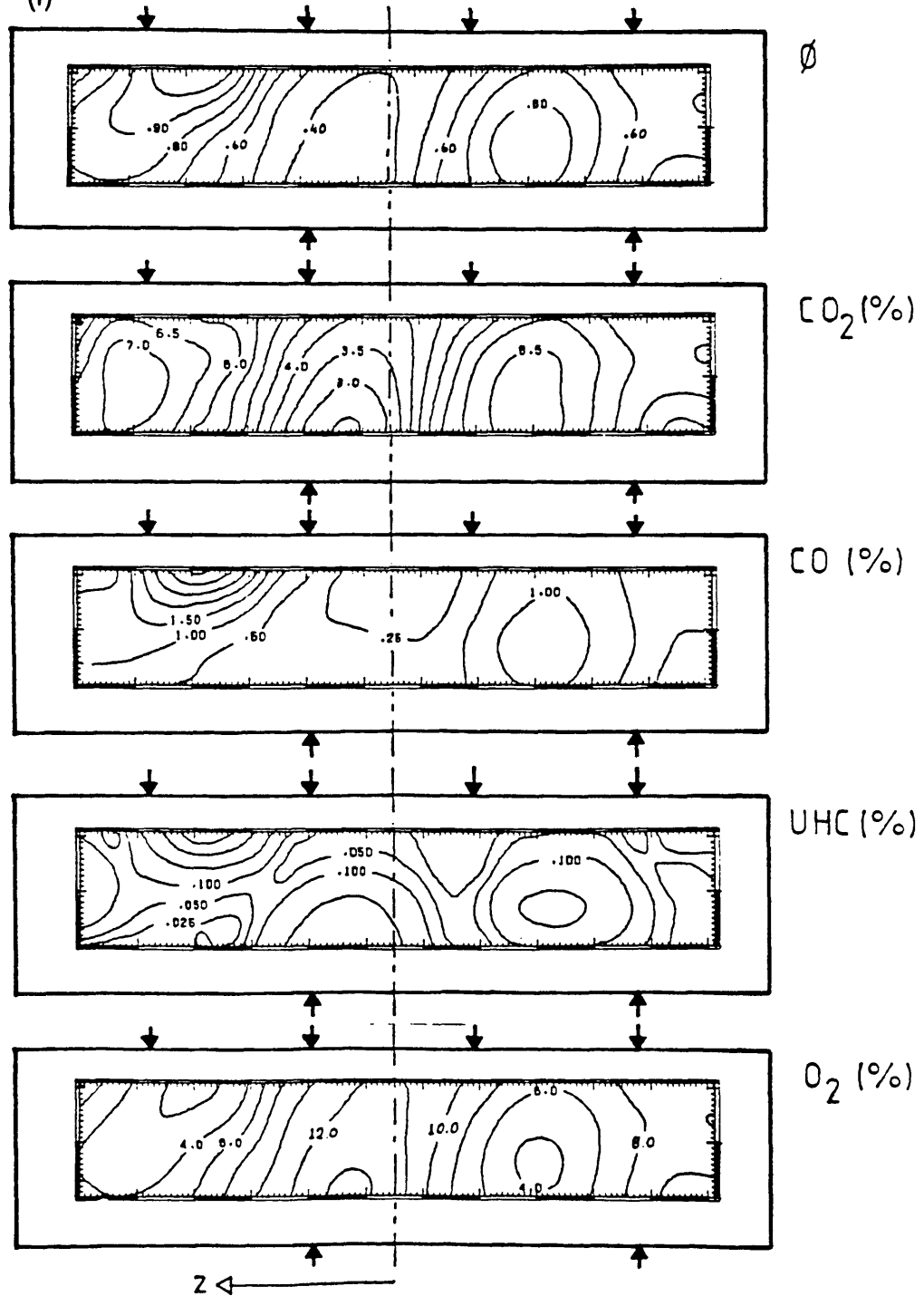
(iii) $T_{inlet}=515K$ and $AFR=29$ in Primary Zone



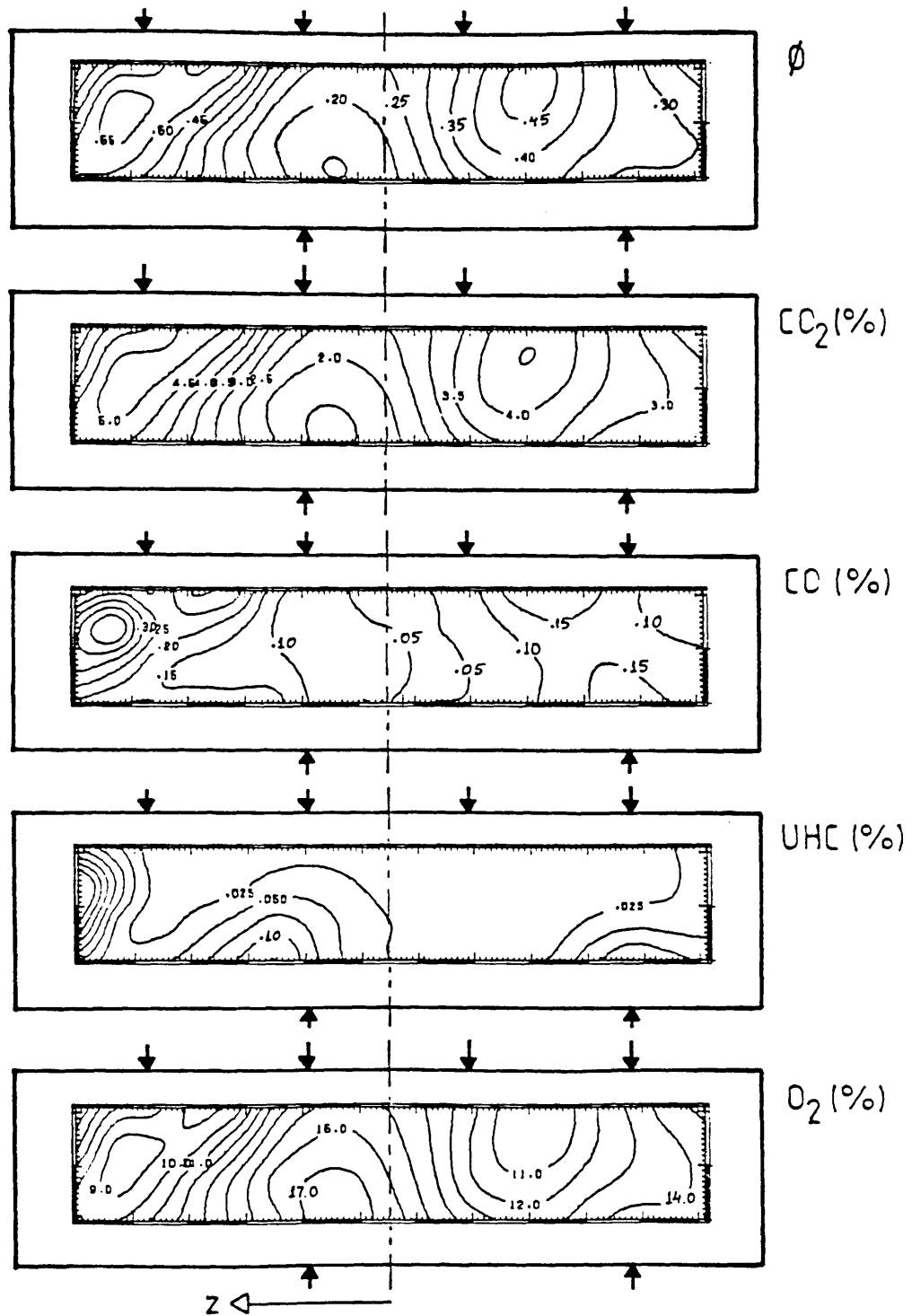
$x = 100 \text{ mm}$

(b)

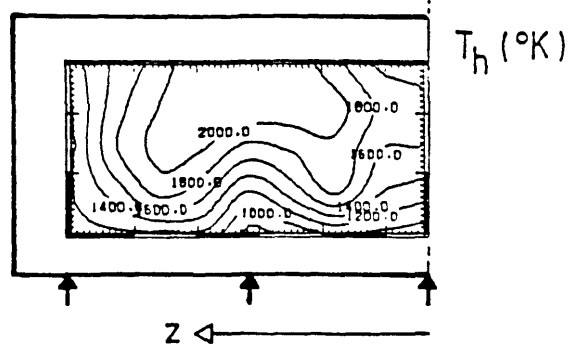
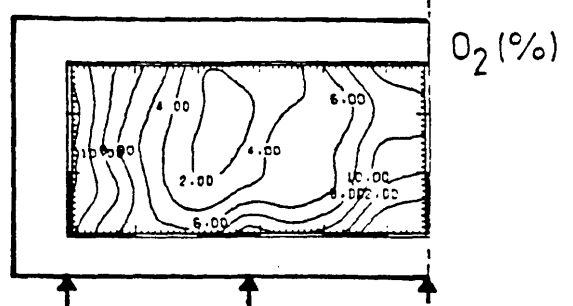
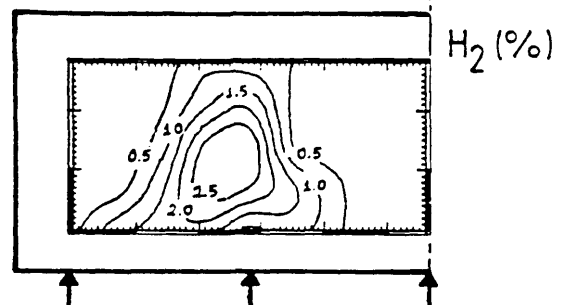
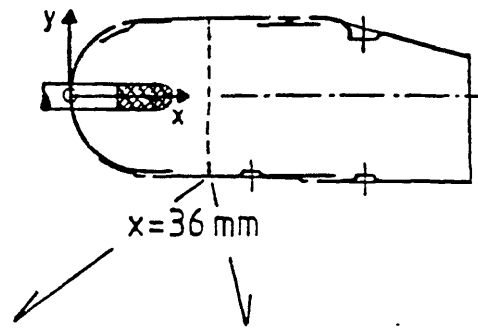
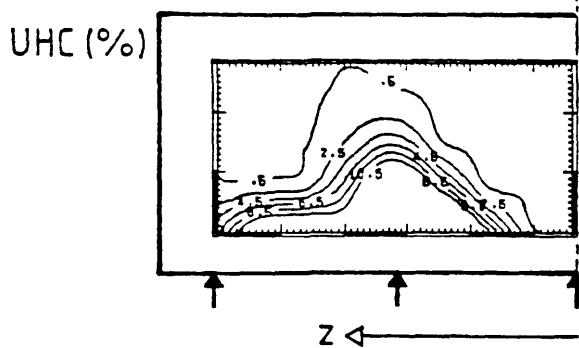
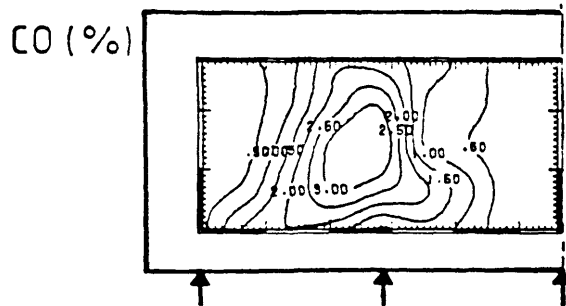
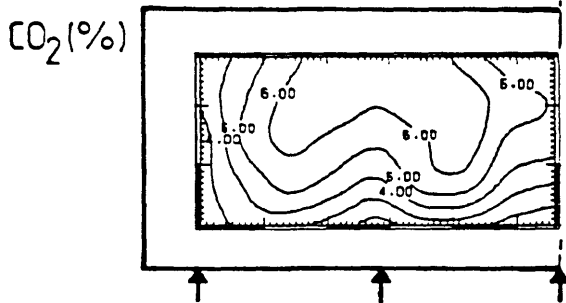
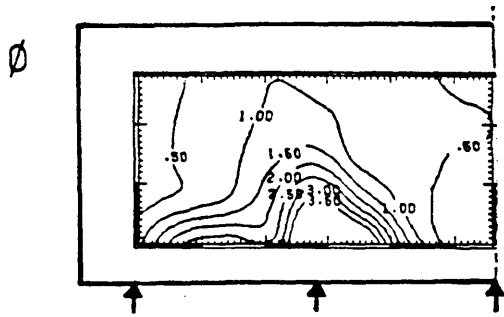
(i)



(ii)



(iii)



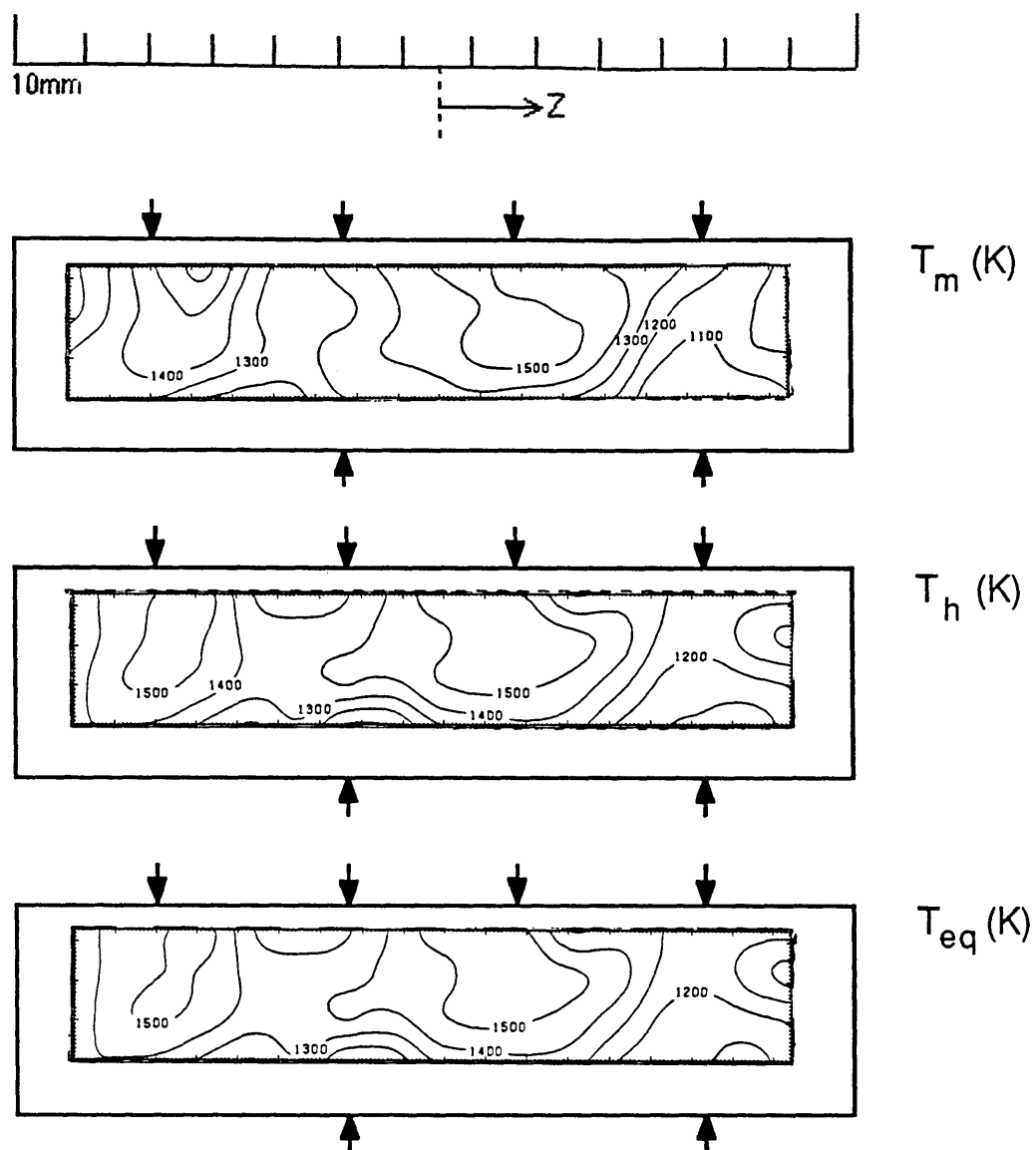


Figure 5.38 Calculated Enthalpy Balance Temperature in Exit Plane for $T_{inlet}=515K$ and $AFR=43$ in Exit Plane

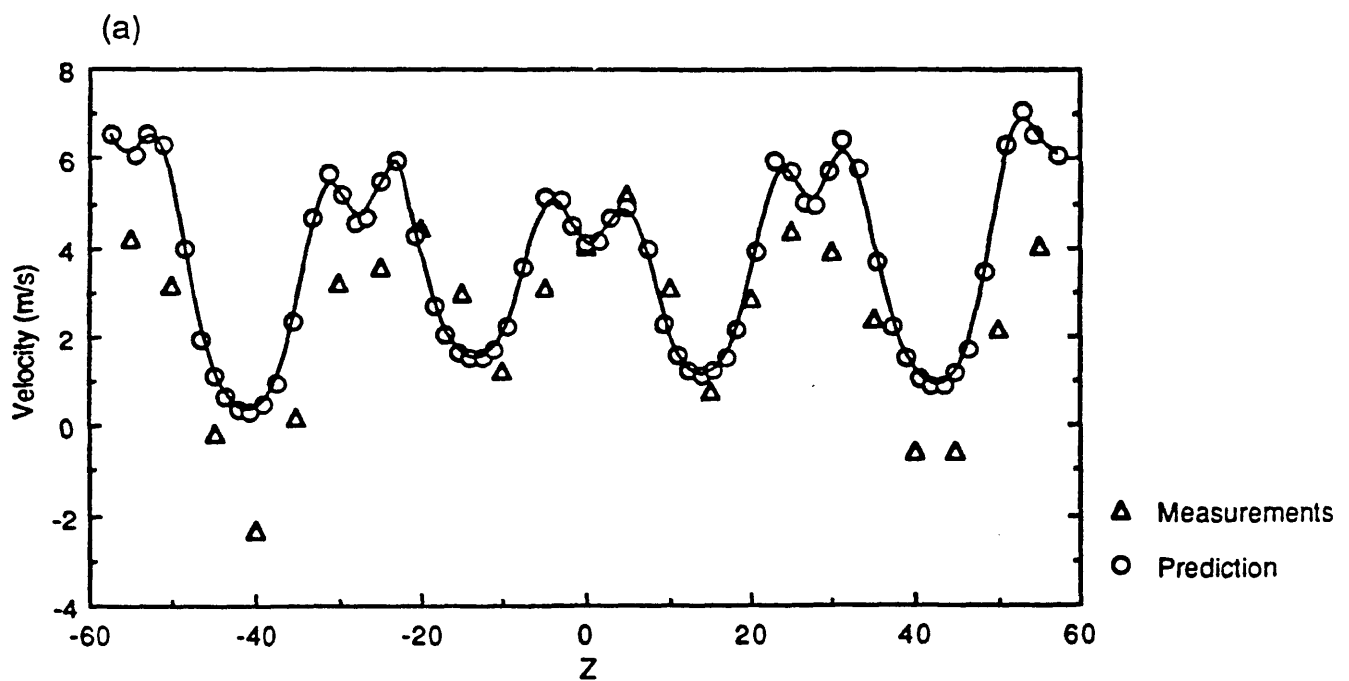


Figure 5.39 Comparison of Velocity Measurements and Prediction in Primary Zone

(a) U-Velocity at $y=0\text{mm}$; $x=45\text{mm}$

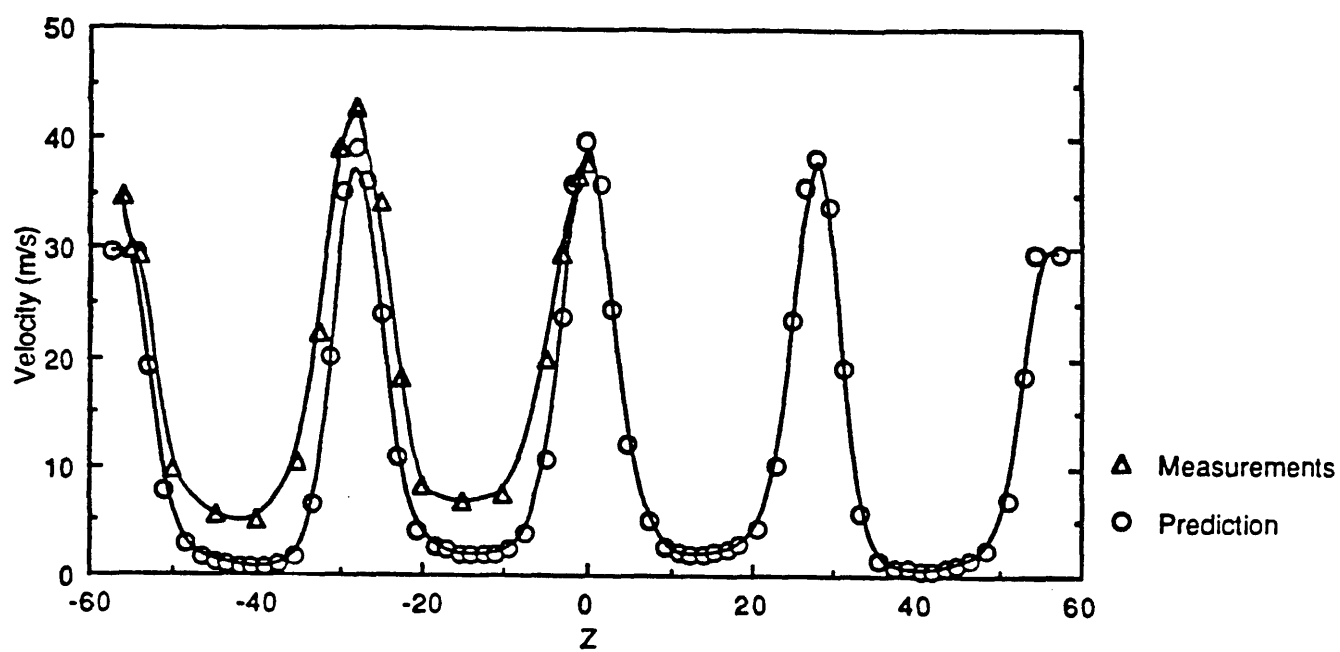
(b) V-Velocity

(i) $y=0\text{mm}$; $x=45\text{mm}$

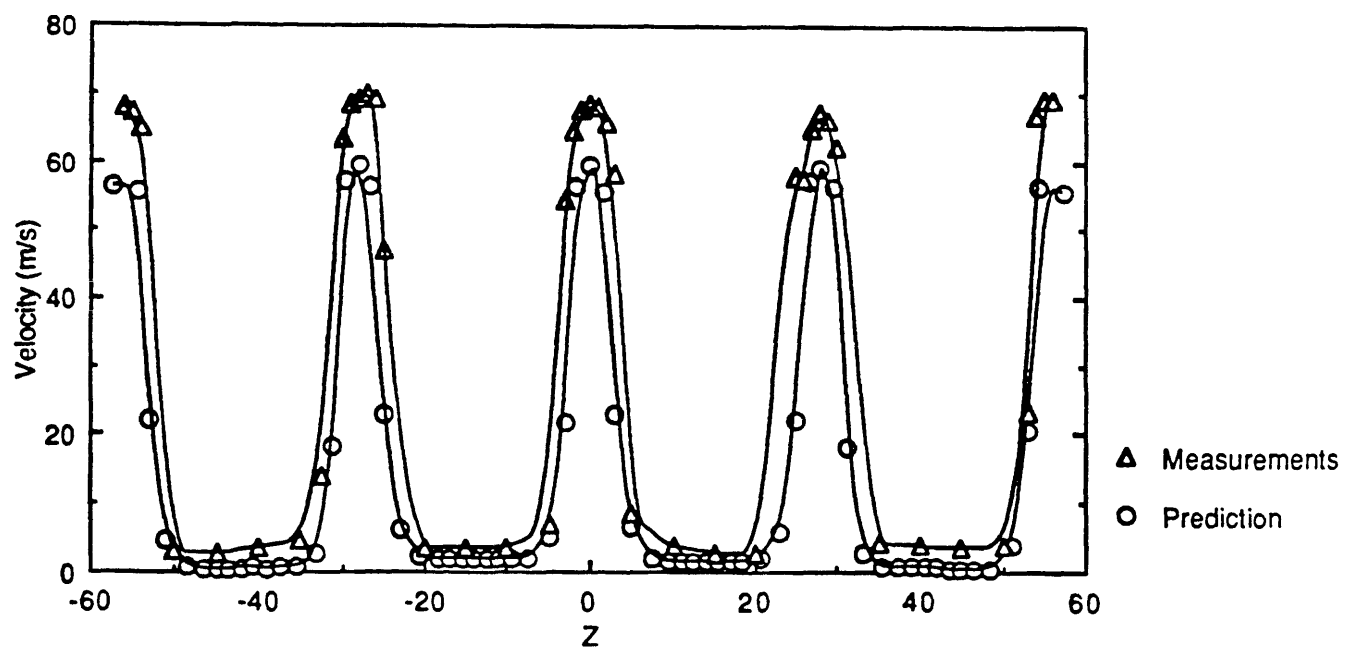
(ii) $y=-10\text{mm}$; $x=45\text{mm}$

(b)

(i)



(ii)



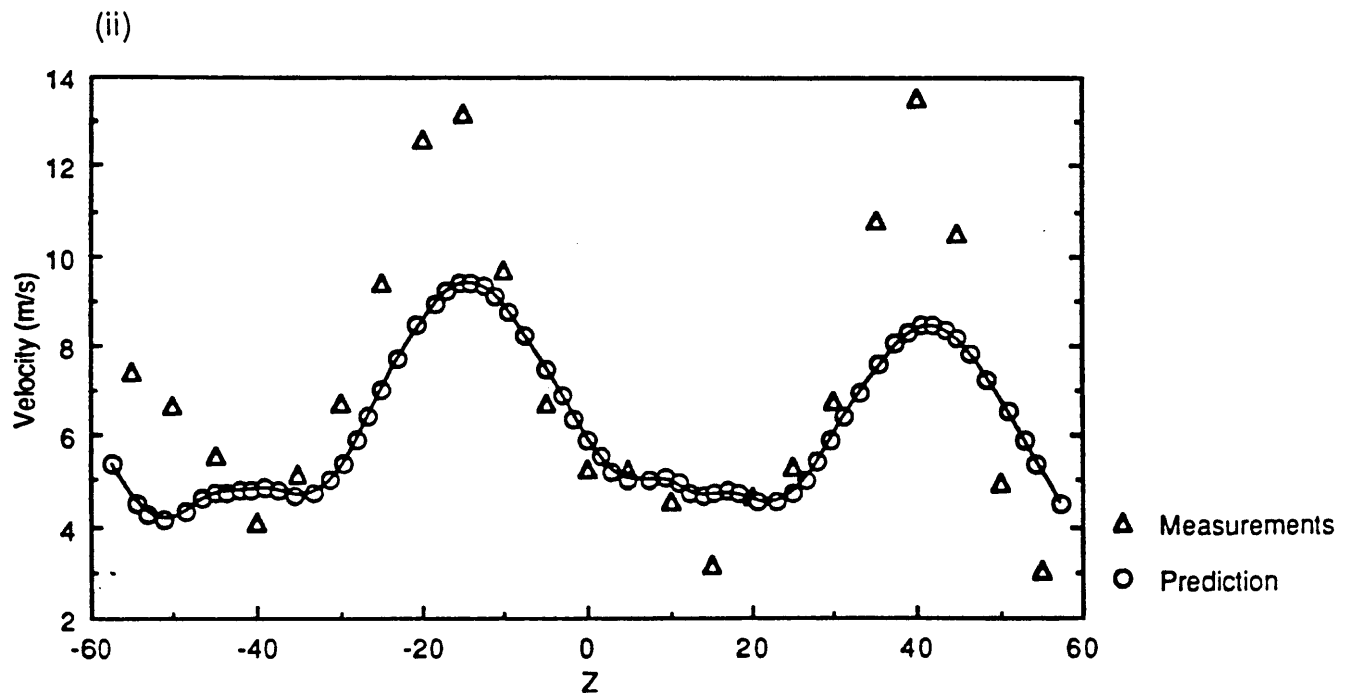


Figure 5.40 Comparison of Velocity Measurement and Prediction in Exit Plane
at $y=0\text{mm}$

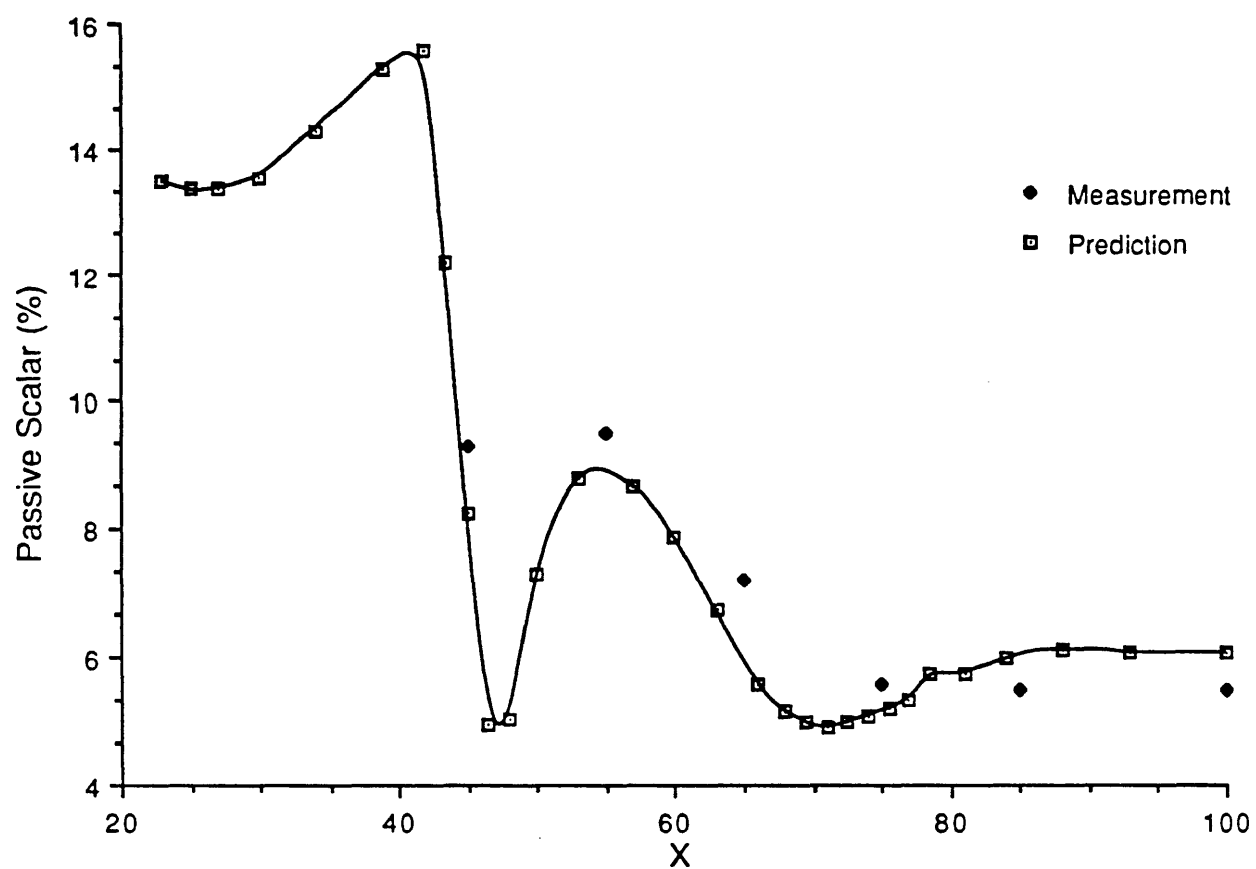


Figure 5.41 Comparison of Passive Scalar Measurements and Prediction at $y=0\text{mm}$ and $z=28\text{mm}$

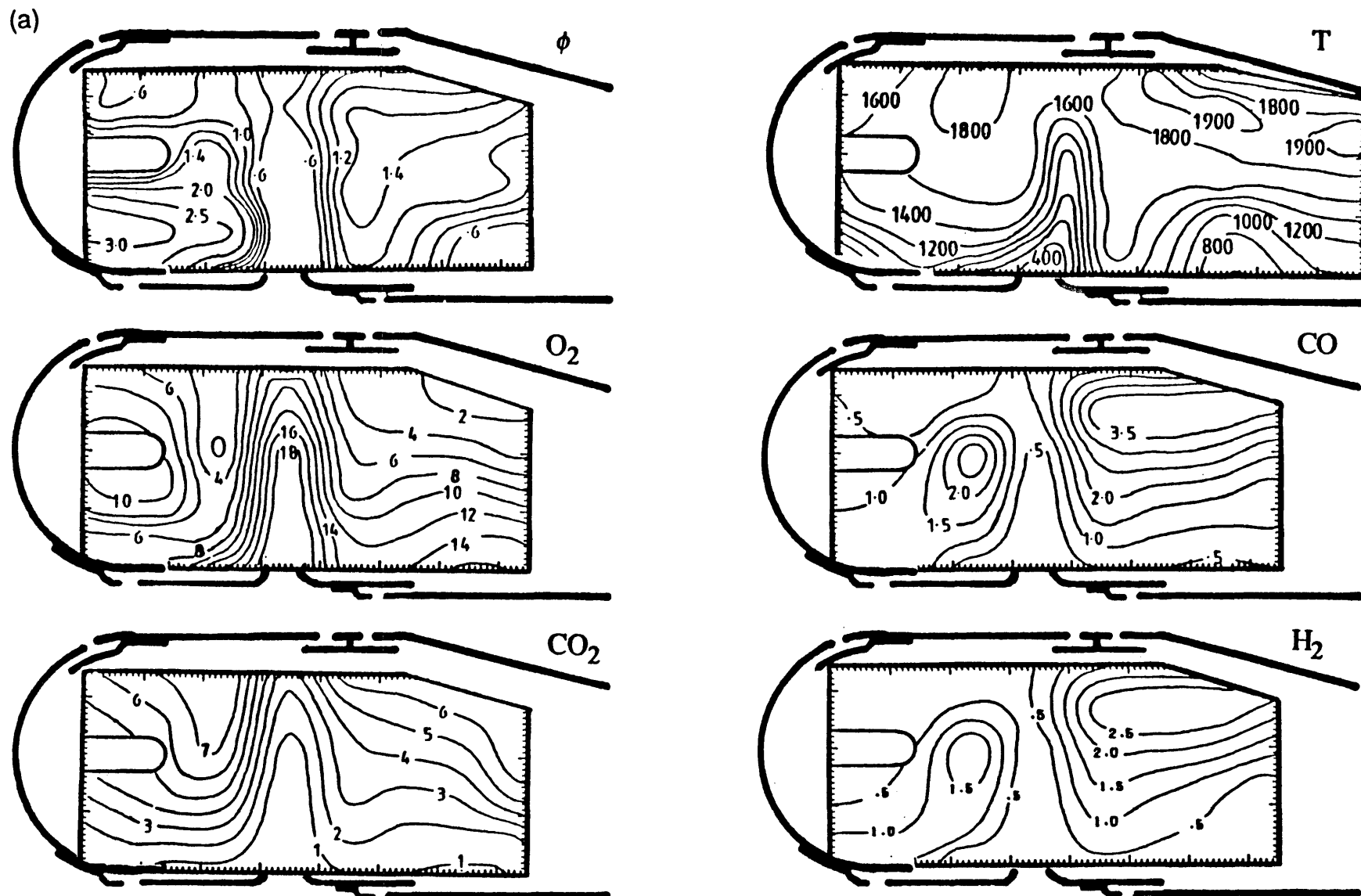
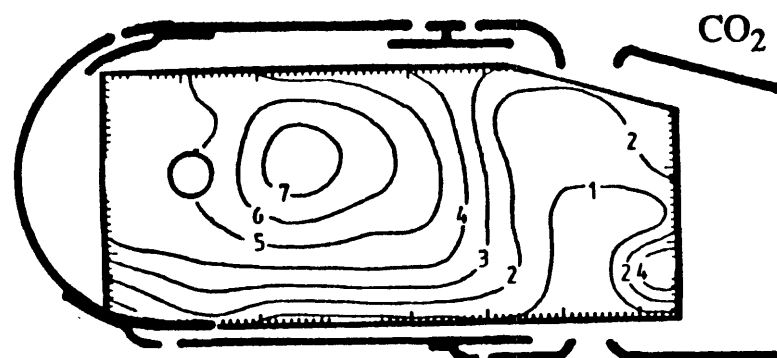
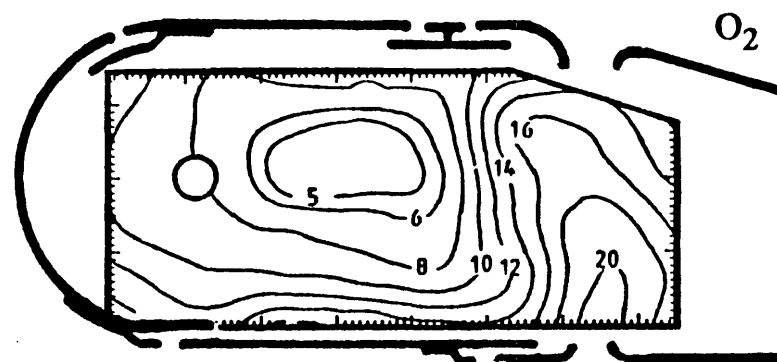
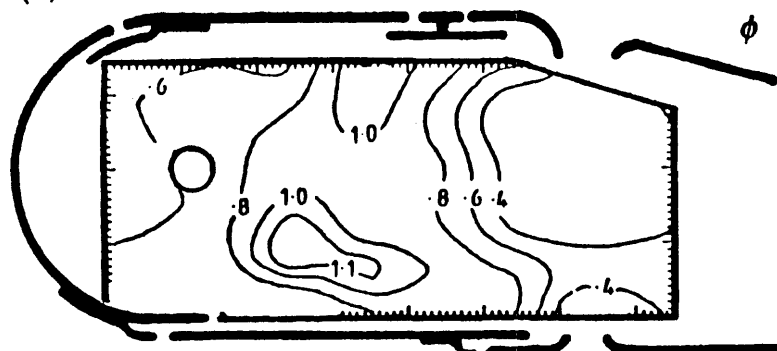
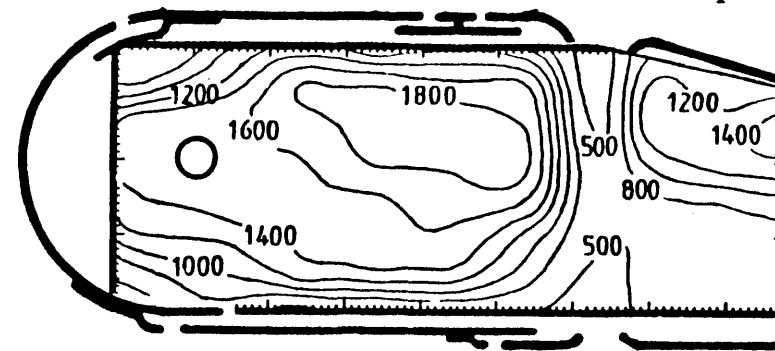


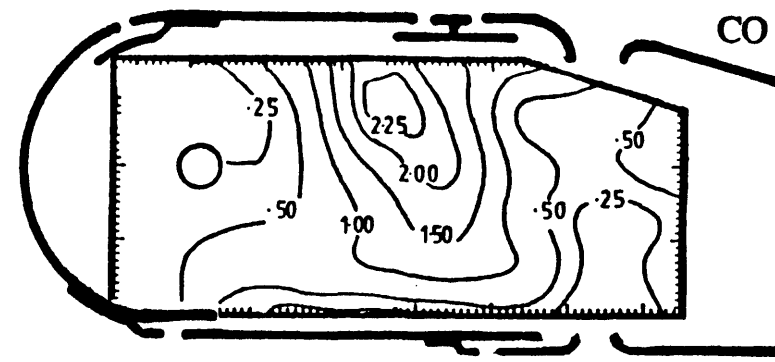
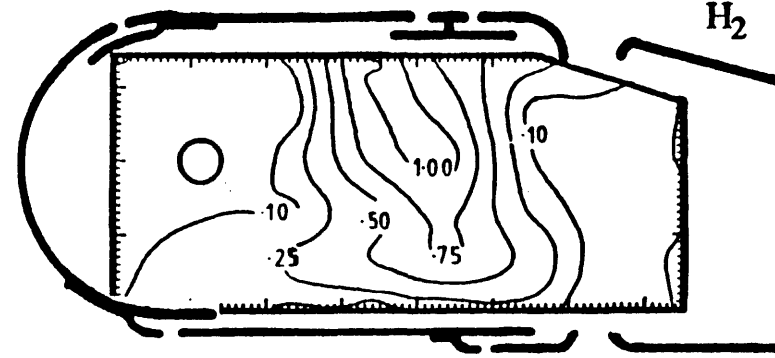
Figure 5.42 Measured Species Concentrations Distributions
 (a) $z = -28\text{mm}$ (b) $z = -14\text{mm}$



T



CO


$$\text{H}_2$$


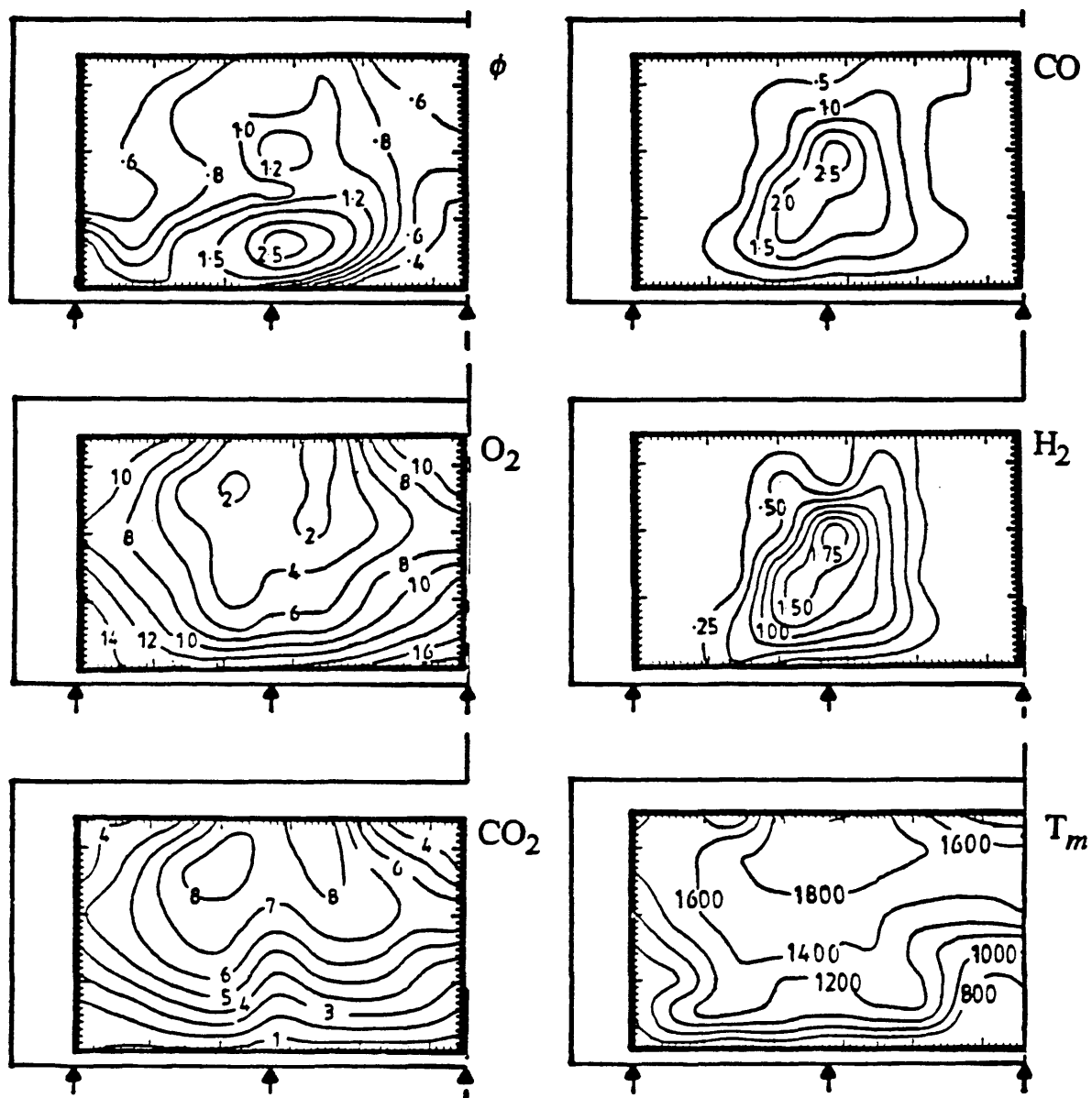


Figure 5.43 Measured Species Concentrations Distributions at $x=36\text{mm}$

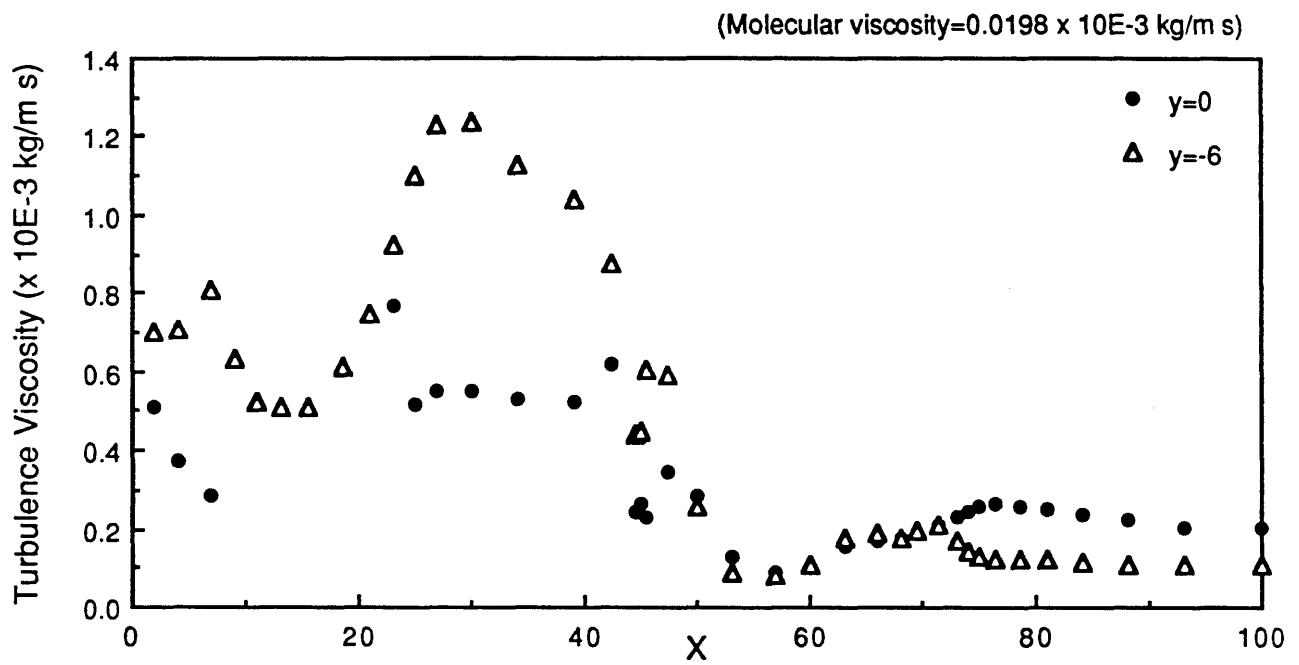


Figure 5.44 Distributions of Calculated Turbulence Viscosity

CHAPTER 6 FLOW AND COMBUSTION CHARACTERISTICS OF A MODEL ANNULAR COMBUSTOR WITH CURVED EXIT NOZZLE

6.1 Introduction

This chapter presents results obtained in a model of a reverse-flow annular combustor with curved exit nozzle. The main objectives of this investigation are to quantify the variation of the flow through the reverse-flow nozzle, with and without combustion, and to provide information of the flow characteristics, including velocity, temperature and species concentrations, in the exit plane of the nozzle. The extent to which the flow in the exit of the combustor is influenced by the curved nozzle can also be examined by comparing results measured with and without the exit nozzle, presented here and in Chapter 5 respectively.

The experimental configuration is similar to that described in Chapter 5 with fuel delivered through a T-vaporiser and air supplied through dilution holes and film-cooling slots via two plenum chambers. The experimental techniques and the associated uncertainties are described in Chapter 2 and laser Doppler velocimetry in forward scattering modes was used to measure the velocity in isothermal and combusting flows, with Kaolin particles as seeding, and the techniques for the measurements of temperatures and species concentrations were identical to those employed in Chapter 5.

The flow configuration is described in Section 6.2 and Section 6.3 presents the measurements of velocity, in isothermal and combusting conditions, species concentrations and temperature. The results are

discussed in Section 6.4 and summary conclusions are provided in the last section. The scalar characteristics at the exit plane of the combustor have been reported by Chow and Whitelaw (1991).

6.2 Flow Configuration

The flow arrangement and the geometry of the model combustor are shown in Figures 6.1 and 6.2 respectively with the geometric details provided in Table 6.1. The fuel supply system was identical to that of Chapter 5 and incorporated an airblast atomiser, Figure 5.2. The only difference in the air supply system, due to the different orientation of the combustor as observed by comparing Figures 5.1 and 6.1, was that the bottom plenum chamber supplied air to the top dilution holes and film-cooling slots at the outer wall of the curved nozzle, and the rest of the air through the top plenum chamber.

The general flow pattern in the primary and dilution zone was described in Chapter 5 and, in the present configuration, four additional film-cooling slots were situated inside the curved nozzle, downstream of the dilution zone, with one at the inside surface, at the entry to the curved nozzle, and three at the outside surface. In the present model, the curved exit nozzle had a trapezium cross-section to maintain the flat top and bottom walls of the previous model and to maintain an area ratio of 5 between the primary zone and the exit of the nozzle, which is that of the actual annular combustor and is essential to the acceleration of the bulk flow within the nozzle. Two quartz windows were mounted on the two sloping side walls to allow optical access for velocity measurements inside the combustor. From the study of the simplified geometry of the curved exit nozzle in Chapter 4, the effect of the side wall was quantified and it

was shown that the secondary flow recirculation vortex was limited in its spanwise extent to less than 30% of the width of the channel and the bulk flow in the central region, which was skewed towards the inner wall as a result of the strong radial pressure gradient, was unaffected by the presence of a side wall. The effect of the secondary flow was also reduced as the flow developed along the duct due to the acceleration caused by the streamwise contraction in cross-sectional area.

The overall air-fuel ratios were set at 36 and 60 corresponding to the take-off and ground-idle conditions respectively and to fuel flow rates of Chapter 5, with the addition of air to the film-cooling slots within the nozzle.

6.3 Results

The results of velocity measurements are presented in Section 6.3.1 which is divided into two parts dealing with isothermal and combusting conditions separately. The velocity measurements made were mainly in the curved exit nozzle of the combustor since previous results of Tse (1988) had already quantified the flow pattern in the primary and dilution zones. Sections 6.3.2 and 6.3.3 deal with the temperature and species concentrations measurements made at the exit of the combustor respectively and the measurement grids is shown in Figure 6.3. The measured concentrations of CO, CO₂ and H₂, on a dry basis, and UHC, on a wet basis, were converted to wet basis in the manner described in Section 5.4.1.

6.3.1 Velocity

(a) Isothermal Flow

The velocity profiles of Figure 6.4, correspond to the entry to the curved exit nozzle in isothermal condition, are non-dimensionalised by the bulk mean velocity (U_b) at the entry plane to the curved nozzle (110mm from the combustor head) with y denoting the distance from the inner curved surface, and shows maxima close to the inner surface of the bend. The averaged velocity across the width of the combustor at $y=5.5\text{mm}$ is $1.25 U_b$ with a maximum of $1.5 U_b$ at $z=-42$ and 14mm corresponding to the planes of dilution-jet impingement. The maxima can still be observed at $y=8$ and 13mm whereas the profiles are considerably dampened and no distinct maximum observed for $y>18\text{mm}$. This indicates that the air from the opposed dilution jets, after impingement, flowed downward towards the inner surface of the combustor. The magnitude of the streamwise velocity decreases as y increases and, at $y=23\text{mm}$, the averaged velocity becomes less than $0.85 U_b$. At $y=28\text{mm}$, the velocity remains the same as that at $y=23\text{mm}$ due to the presence of the cooling slot at the top wall of the combustor near the entry to the curved bend where the flow near the top wall was accelerated by the faster moving air from the slot. Therefore, the effect of the radial pressure gradient, caused a lower velocity as y increased, near the top wall was cancelled by the acceleration caused by the air flow from the cooling slot, which accounted for about 10% of the total air flow to the combustor and led to a slot exit velocity of about $2.0 U_b$. The profiles of the rms of velocity fluctuations are relatively uniform at around $0.35 U_b$, since most of the turbulence energy has been produced upstream of this plane by the various air jets and their interaction with the bulk flow as suggested by Tse(1988).

Figure 6.5 shows the streamwise velocity profiles at the centre plane, $z=0\text{mm}$, for further downstream regions. At the 45° plane, the maximum velocity remains close to the inner wall with magnitude of $2.0 U_b$. The effect of the second cooling slot at the outer surface can be observed when comparing the velocities at $y=17.5$ and 20mm . As in the case of the entry plane, the slower moving air close to the outer wall was accelerated by the air coming out of the cooling slot and counteracted the effect of the radial pressure gradient.

At the 90° plane, the velocity profile has its maximum near the inner wall with a value of $2.5 U_b$ at $y=2.5\text{mm}$ and the magnitude of the velocity decreases to less than $2.0 U_b$ at $y=17.5\text{mm}$. The rms values of the velocity fluctuations are uniform across the whole plane with a value of about $0.25 U_b$, and are comparatively low due to the acceleration of the bulk flow. The same trend is observed in the 135° plane with velocity maximum and minimum of $3.5 U_b$ and $2.0 U_b$ near the inner and outer wall respectively. The accelerating effect of the third cooling slot at the outer surface of the nozzle is not obvious when comparing the velocities at $y=10.5$ and 14.5mm in the 135° plane since the bulk mean velocity at this plane is already high and although the cooling slot supplied extra mass flux at the outer surface, the effect was less than that of the two cooling slots at further upstream regions.

At the 180° plane, the difference in the velocity distribution diminishes with maximum and minimum of $3.6 U_b$ and $3.2 U_b$ respectively due to the absence of a strong radial pressure gradient as in further upstream regions. At the exit of the combustor, a uniformly distributed velocity profile, as in the case of the bend flow in Chapter 4, is observed with an average velocity of $3.5 U_b$.

Figures 6.6 to 6.8 show the radial velocity profiles at entry to the bend, the 90° plane and at the exit of the combustor where negative values denotes velocity toward the inner surface. A recirculation region is observed in the entry and 90° planes but is limited to about 20mm in the spanwise direction. At the exit, the velocity vector is towards the inner surface and decreases in magnitude as y increases with the air flowing normal out of the exit plane for $y > 7.5\text{mm}$. This is because the flow out of the exit at the inner surface resembled the flow through a forward facing step and the fluid reattached downstream of the plenum chamber. A small recirculation region was found in the corner of the plenum chamber and the low pressure caused the inward flow of air near the inner surface of the combustor exit.

The general nature of the flow through the exit nozzle was similar to that in the idealised duct studied in Chapter 4 whereas the influences of the non-uniform inlet velocity profile and film-cooling slots at the two curved surfaces can be observed from the results presented above. The differences between the two flows diminished along the ducts and at the exit, uniform streamwise velocity distributions were observed in both cases indicates that the convergence in streamwise cross-sectional area had a more significant effect to the flow at downstream locations.

(b) Combusting Flow

Velocity measurements were made for combusting flow with an air-fuel ratio of 60 corresponding to ground-idle condition. Figure 6.9 shows the streamwise velocity profiles, non-dimensionalised by the bulk mean velocity in the combusting condition (U_c) at the entry to the curved nozzle. As for isothermal flow, the streamwise velocity profile has its maximum near the inner surface of the bend and two distinct velocity maxima are

observed at $z=-42$ and 14mm , corresponding to the planes of dilution jets impingement. The profile is considerably dampened in the top half of the combustor at $y>18\text{mm}$. The absolute magnitudes of the rms of velocity fluctuations are almost double than those in isothermal flow due to the extra turbulence energy generated by combustion in upstream regions. Streamwise velocity in further downstream region, Figure 6.10, shows that the distributions are more uniform with a maximum of $2.4 U_c$ at $y=2.5\text{mm}$ and minimum velocity of $1.8 U_c$ at $y=17.5\text{mm}$ in the 90° plane. At the combustor exit, the velocities across the whole exit plane are between $3.0 U_c$ and $3.5 U_c$ indicating a very uniform profile.

Comparing results obtained in isothermal and combustive conditions, it is evident that the flow developments inside the curved nozzle, in non-dimensional form, are similar in both cases, taking into account the difference in velocity profiles at the entry to the nozzle due to combustion. The strong curvature of the nozzle created a strong radial pressure gradient and hence the velocity maxima remained close to the inner surface. The close similarity of the velocity distributions within the nozzle in the two cases also suggests that the momentum transfer by diffusion caused by temperature gradient, in the cross-stream plane, is negligible in the present configuration.

6.3.2 Temperature

The distribution of measured temperature, and the corresponding enthalpy balance temperature, are shown in Figures 6.11 and 6.12 for air-fuel ratios of 60 and 36 respectively. In both cases, high temperature regions are close to the inner wall of the curved exit nozzle with maximum of 1350K and 1650K for the two air-fuel ratios. The enthalpy balance temperatures are generally about 50 to 100K lower than those measured, in

accordance with the finding of Section 5.5.3. The calculated pattern factors, based on 78% of the area of the exit plane, are 0.22 and 0.18 for air-fuel ratio of 60 and 36 respectively. These values are lower than those obtained in the model combustor of Chapter 5 due to the absence of a distinct high temperature region.

6.3.3 Species Concentrations

Figure 6.13 shows the distributions of major species concentrations, on a wet basis, and calculated equivalence ratios for the air-fuel ratio of 60. Maximum equivalence ratios above 0.3 are found close to the inner wall of the curved bend and the equivalence ratios at the outer bend are generally about 0.2. No distinct hot zone is observed across the width of the exit plane and maximum levels of UHC and CO are 0.006 (mole fraction of $C_{12}H_{26}$) and 0.15% respectively. This corresponds to emission indices of CO and UHC of 50 and 5 respectively which implies a combustion efficiency of 98.3%.

The distributions of species concentrations for the air-fuel ratio of 36 are shown in Figure 6.14. The fuel rich regions have maximum equivalence ratios of 0.48 and remain close to the inner wall, as in the case of the higher air-fuel ratio, and the maximum levels of UHC and CO increase to 0.009 and 0.18% respectively. Compared to the higher air-fuel ratio condition, the emission indices of CO and UHC increase to 68 and 6 respectively and a slight decrease in combustion efficiency to 97.8% resulted. The equilibrium calculations in Chapter 5 at the exit of the model without curved exit nozzle, which was the entry plane to the curved nozzle in the combustor in this chapter, showed that the reaction had already proceeded to an equilibrium state and therefore the equilibrium calculation was not repeated in the exit of the combustor with exit nozzle. The

performance of the combustor with the curved exit nozzle in terms of combustion efficiency, emission indices of CO and UHC and pattern factor is also shown in Table 6.2. The results presented in this chapter are compared, in the next section, with the scalar measurements presented in Chapter 5 and previous velocity measurements by Tse (1988) to illustrate the effect of the curved nozzle to the velocity and scalar distribution within the nozzle and at the exit of the combustor..

6.4 Discussion

The velocity measurements at the entry to the curved nozzle of this chapter can be compared with those obtained previously, by Tse (1988), without the exit nozzle, to quantify the upstream influence of the curved nozzle. However, due to the different arrangement of the plenum chamber for the air-feed, velocity measurements were made in the present geometry to quantify the difference in velocity profiles in upstream regions prior to the comparison. The streamwise velocity profiles at $x=60\text{mm}$, Figure 6.15, suggest that the flow pattern in upstream regions is unaffected despite the difference in method of air supply and there is no upstream influence of the curved geometry observed in this location which is 50mm from the entry to the curved section of the combustor. The absence of upstream influence in this location agrees with the results presented in Chapter 4 for which no effect of the curvature was observed at a position 54mm upstream of the entry to the curved duct with identical curvature.

Figure 6.16 shows the velocity measurements made at $x=110\text{mm}$ in the combustor without the curved nozzle and the two velocity maxima, correspond to the dilution jet impingement are observed, as in Figure 6.4. The contraction towards the entry of the nozzle has not accelerated the

flow in the upper regions in either case. Instead, the flow has been pushed downward by the radial pressure gradient, caused by the curved nozzle, and leads to more distinct velocity maxima near the inner wall of the combustor as shown in Figure 6.4, whereas in Figure 6.16 this region of distinct velocity maxima is about 10 to 15mm above the inner wall.

The distributions of species concentrations at the exit of the curved nozzle of Figures 6.13 and 6.14 for the two air-fuel ratios can be compared with those at the exit of the combustor without the nozzle in Figures 5.7 and 5.11 to quantify the effect of the nozzle on the combustion processes. For the ground-idle condition, the maximum value of equivalence ratio has decreased from 0.42 to 0.3 due to the additional air from the cooling slots inside the curved nozzle. The two distinct hot zones at the entry are absent at the exit due to the intense mixing within the nozzle caused by the severe contraction in cross-sectional area (the area ratio between the entry and the exit is 4.25). The high concentrations of fuel at the outer half of the combustor, at the entry, are pushed to regions near the inner wall at the exit due, in part, to the secondary flow inside the bend which led to an inward radial motion toward the inner surface. Moreover, the fluid near the outer wall was diluted further by the air coming out from the three film-cooling slots at the outer wall of the nozzle.

The average concentrations of UHC and CO have dropped from 0.009 to 0.006% and from 0.18 to 0.12% respectively within the bend at the ground-idle condition. Taking into account the dilution by the extra air from the cooling slots within the bend, which was about 40% of that coming into the combustor at upstream regions, the absolute amount of CO and UHC decreased by a small amount inside the bend and it can be concluded that the fuel and CO burn-up process ceased downstream of the entry of the bend. This is because the temperature in regions of high

concentrations of CO and UHC was below 1400K, Figure 5.6, which is the kinetic limit for fuel and CO burn-up rate as suggested by Lefebvre (1983).

For the take-off condition, the average concentration of UHC has dropped from 0.04 to 0.007% and that of CO from 0.8 to 0.15%. Contrary to the higher air-fuel ratio condition, the combustion processes continued beyond the entry of the curved bend and the fuel breakup and CO to CO₂ conversion took place as a result of the higher local temperature, about 1800K as shown in Figure 5.13, at that plane since the kinetic limitation of the fuel and CO burn-up were overcome.

6.5 Conclusions

This chapter presents velocity measurements, in isothermal and combustion conditions, inside the curved exit nozzle in a model of the reverse-flow annular combustor which incorporates a T-vaporiser for fuel supply, three sets of dilution jets, nine film-cooling slots and a curved exit nozzle. In combusting flow conditions, temperature and species concentrations measurements were obtained at the exit of the nozzle where the combustor operated at near atmospheric pressure, an inlet temperature of 515K and air-fuel ratios of 36 and 60. The main conclusions can be summarised by the following paragraphs:

1. For isothermal and combusting flows, the flow developments within the curved nozzle were similar, with differences less than 13% in the non-dimensionalised velocity profiles. Uniform velocity distributions, within 10% of the bulk mean velocity, were observed at the nozzle exit, despite

the highly non-uniform inlet velocity profiles to the entry plane, and because of the severe contraction in cross-sectional area.

2. Compared with velocity measurements obtained in an identical configuration but without the curved exit nozzle, the streamwise velocity distributions at the entry to the curved exit nozzle were skewed towards the inner wall of the combustor due to strong radial pressure gradient caused by the geometry of the nozzle.

3. For take-off condition, air-fuel ratio of 36, combustion processes continued beyond the entry to the curved nozzle and led to lower emission indices of CO and UHC and a higher combustion of 97.8% at the exit of the nozzle, as compared to the entry of the nozzle.

4. An increase in air-fuel ratio from 36 to 60 resulted in an increase in pattern factor from 0.18 to 0.22 and in combustion efficiency from 97.8% to 98.3% at the nozzle exit.

Table 6.1 Geometric Details of Combustor

Cooling Slots	Slot Height/mm	No. of Holes	Diameter/mm	Pitch/mm
2	1.65	32	2.25	5.91
3	1.38	32	2.00	5.20
4	1.38	28	1.65	5.20
5	1.60	18	2.40	6.84
7	1.10	18	1.65	6.84
9	1.00	15	1.65	6.84
10	1.00	28	1.65	5.20
11	1.00	24	1.65	5.20
12	1.00	18	1.65	5.20

Primary and Dilution Holes	Hole Diameter/mm	No. of Hole	Pitch/mm
1	6.2	4	28
6	4.6	5	28
8	4.6	2	56

Table 6.2 Performance of the Combustor

Fuel	Air-Fuel Ratio	Pattern Factor	El _{CO} (g/kg fuel)	El _{UHC} (g/kg fuel)	Combustion Efficiency (%)
Kerosene	36	0.18	68	6	97.8
	60	0.22	50	5	98.3

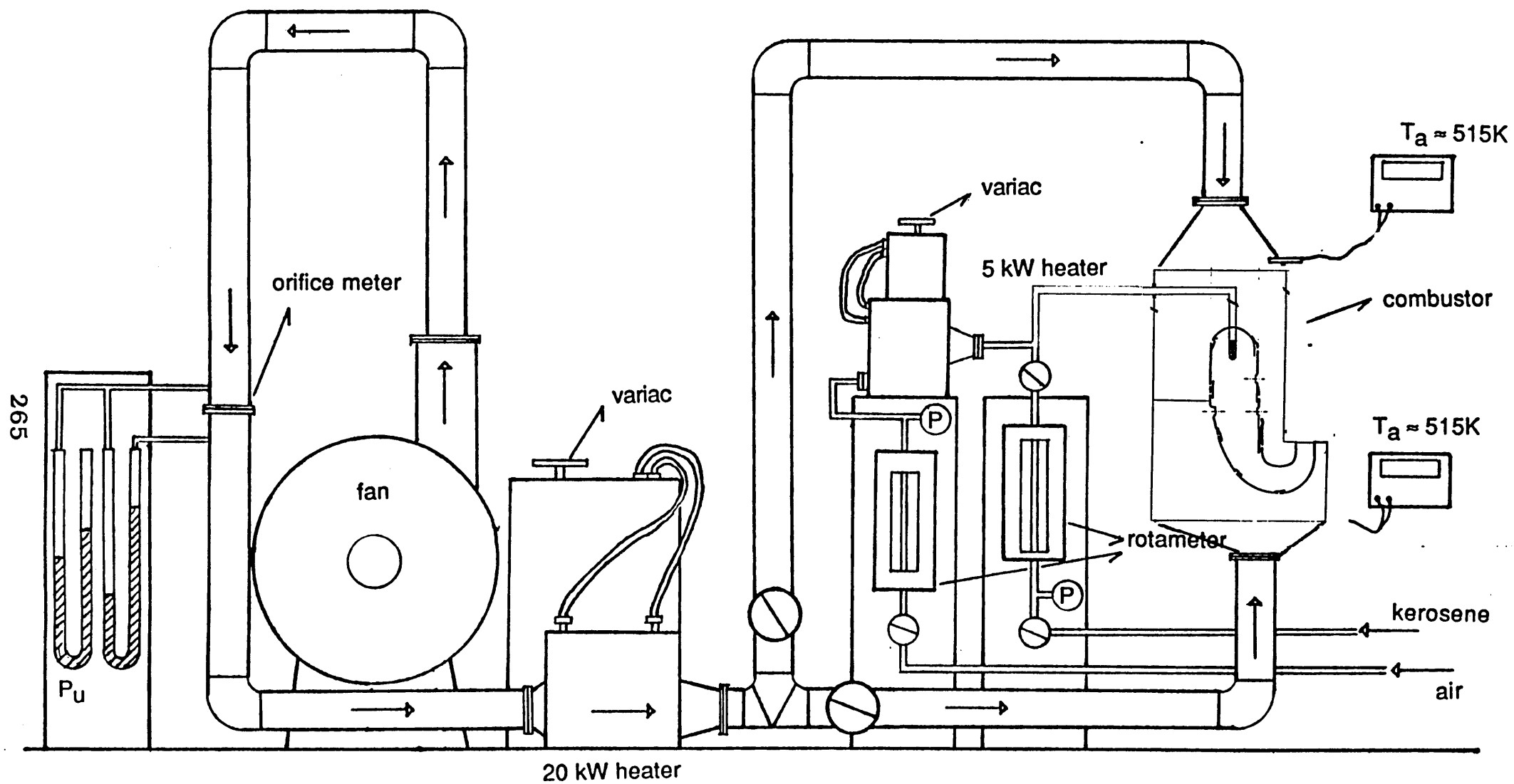


Figure 6.1 Flow Arrangement

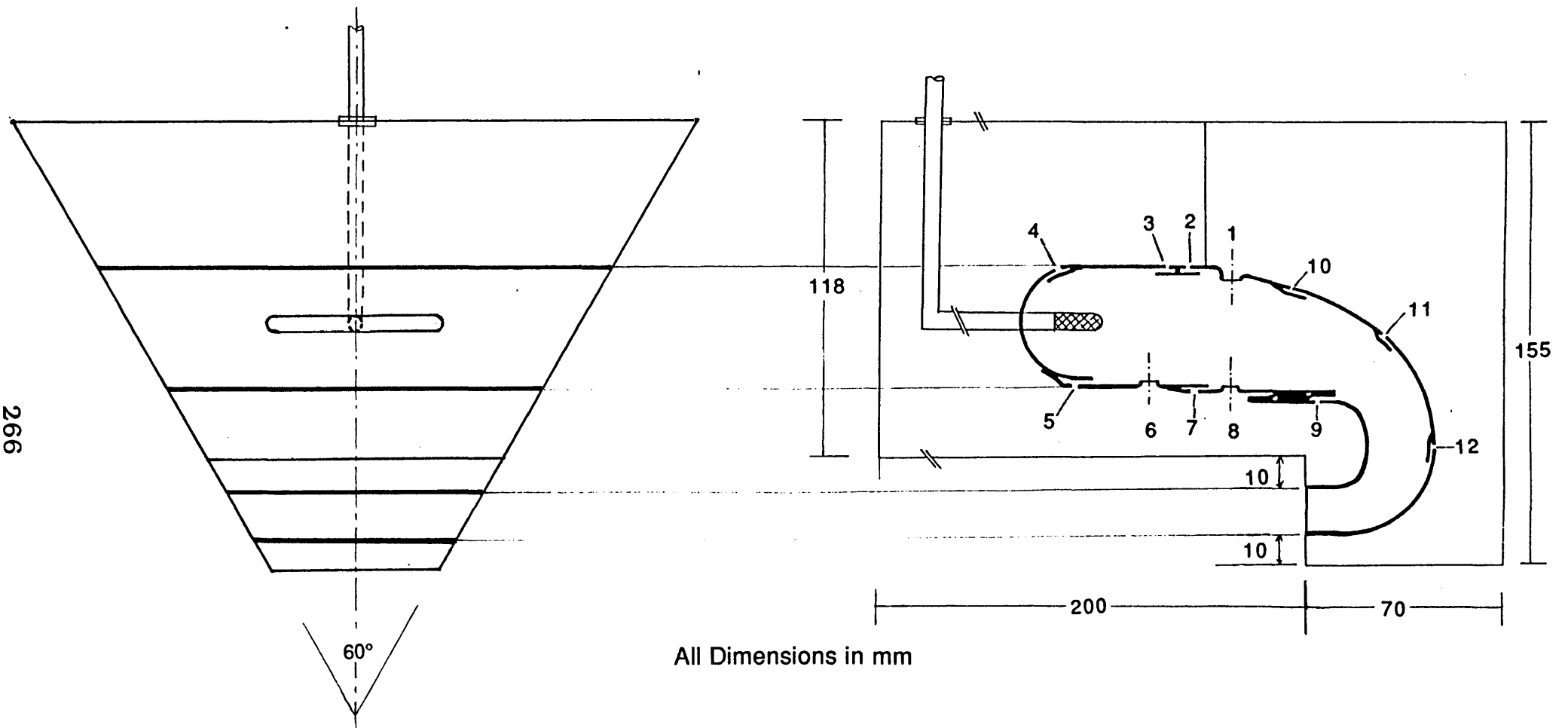


Figure 6.2 Combustor Geometry

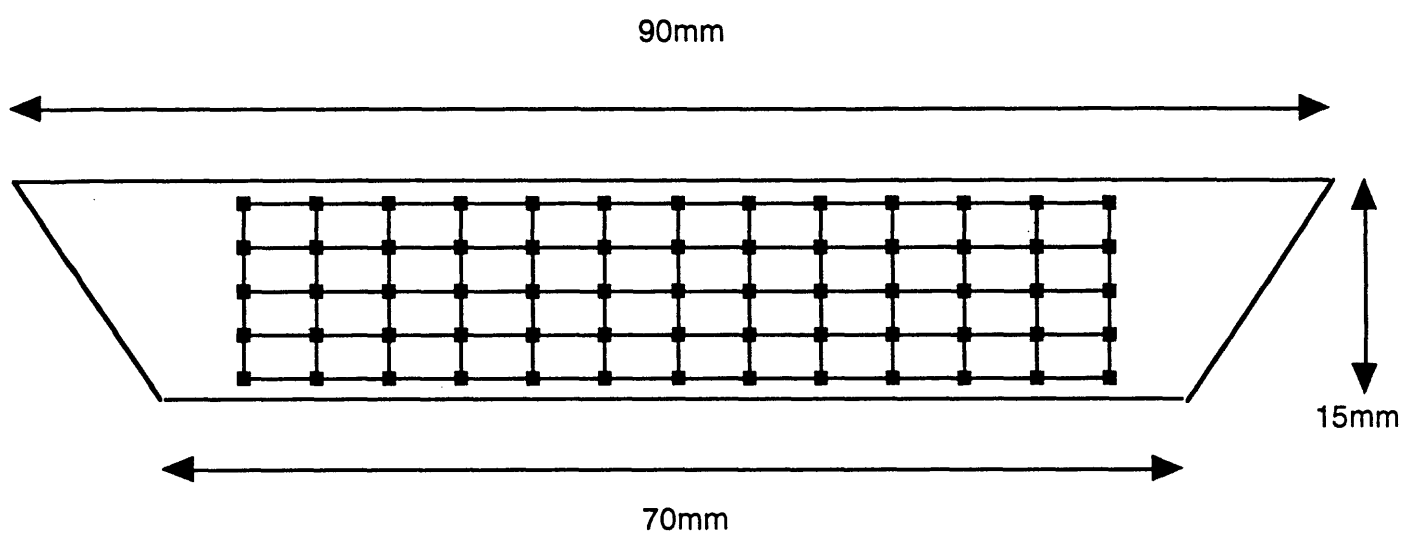


Figure 6.3 Measurement Grid in Exit Plane of the Combustor

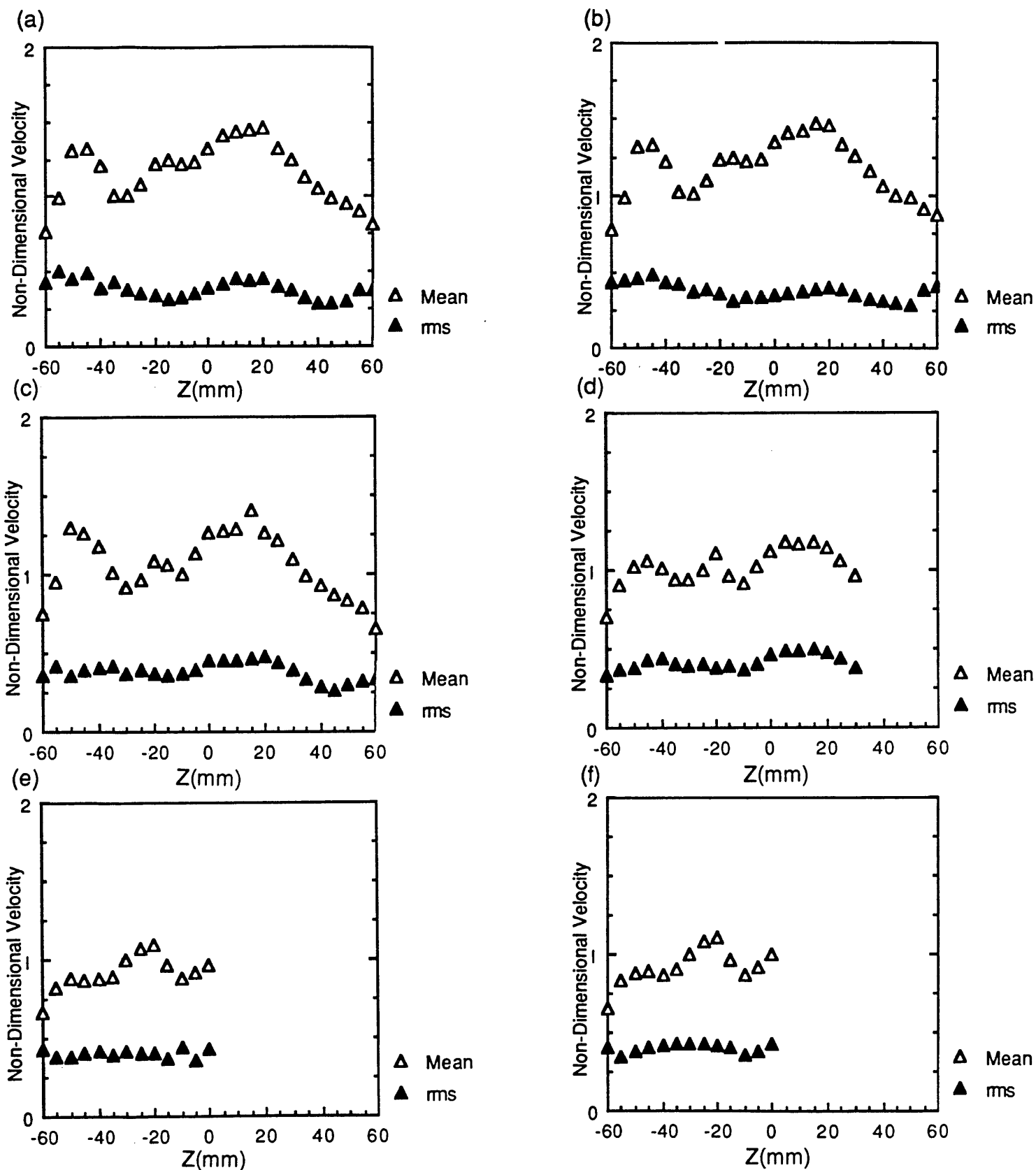


Figure 6.4 Profiles of Measured Mean and rms Streamwise Velocity at Entry to

Curved Nozzle in Isothermal Flow

(a) $y=5.5\text{mm}$

(b) $y=8\text{mm}$

(c) $y=13\text{mm}$

(d) $y=18\text{mm}$

(e) $y=23\text{mm}$

(f) $y=28\text{mm}$

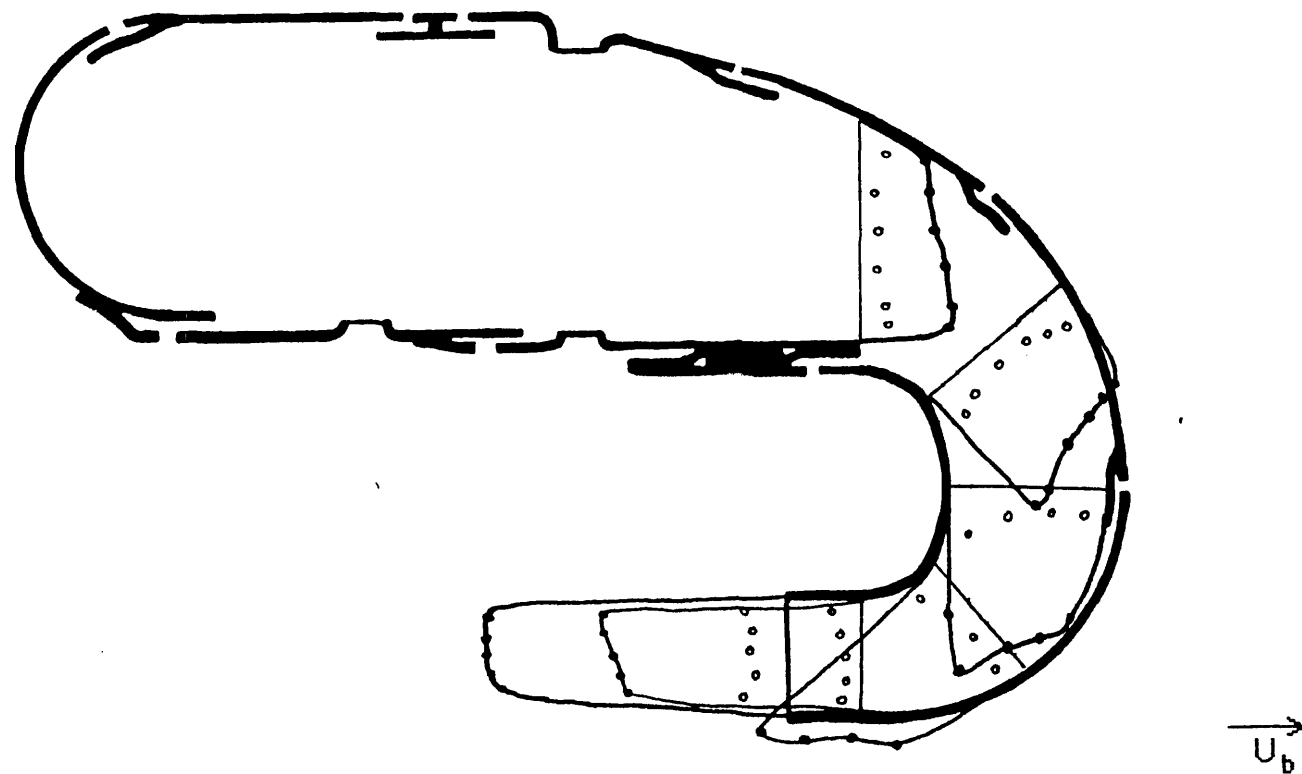


Figure 6.5 Profiles of Measured Mean and rms Streamwise Velocity at $z=0\text{mm}$ in Isothermal Flow

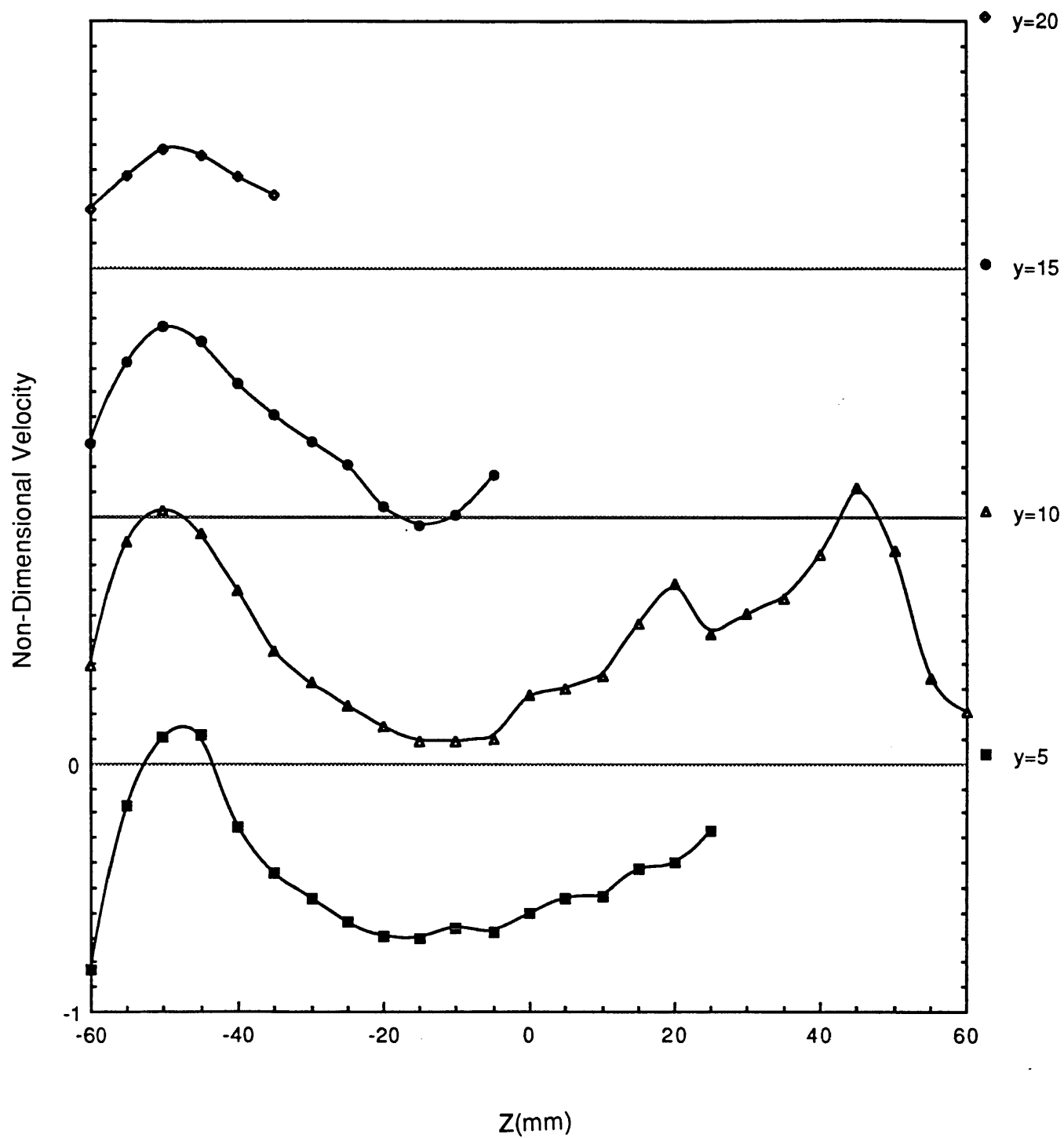


Figure 6.6 Profiles of Measured Radial Velocity at Entry to Curved Nozzle in Isothermal Flow

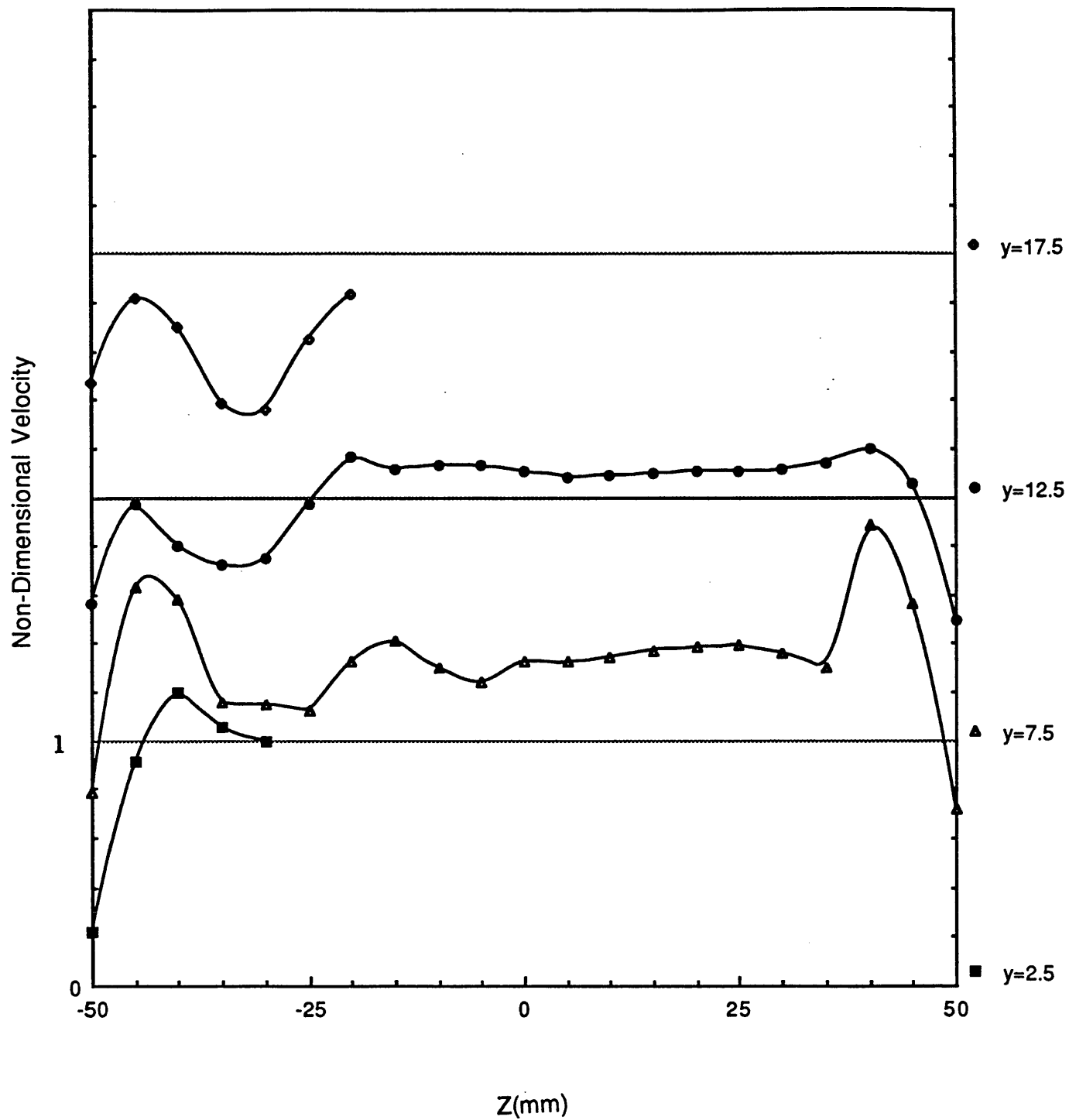


Figure 6.7 Profiles of Measured Radial Velocity in 90° Plane in Isothermal Flow

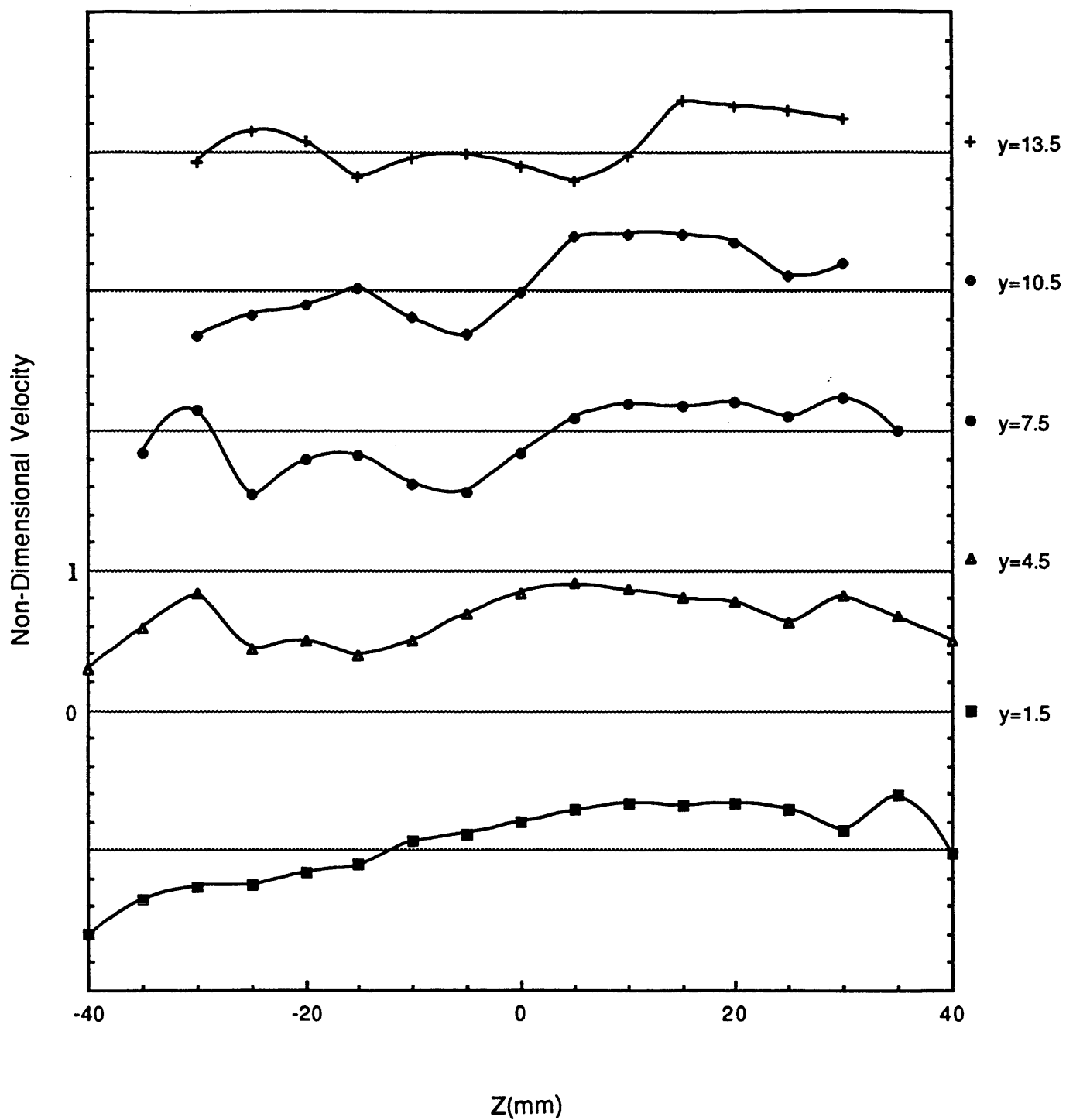


Figure 6.8 Profiles of Measured Radial Velocity in Exit Plane in Isothermal Flow

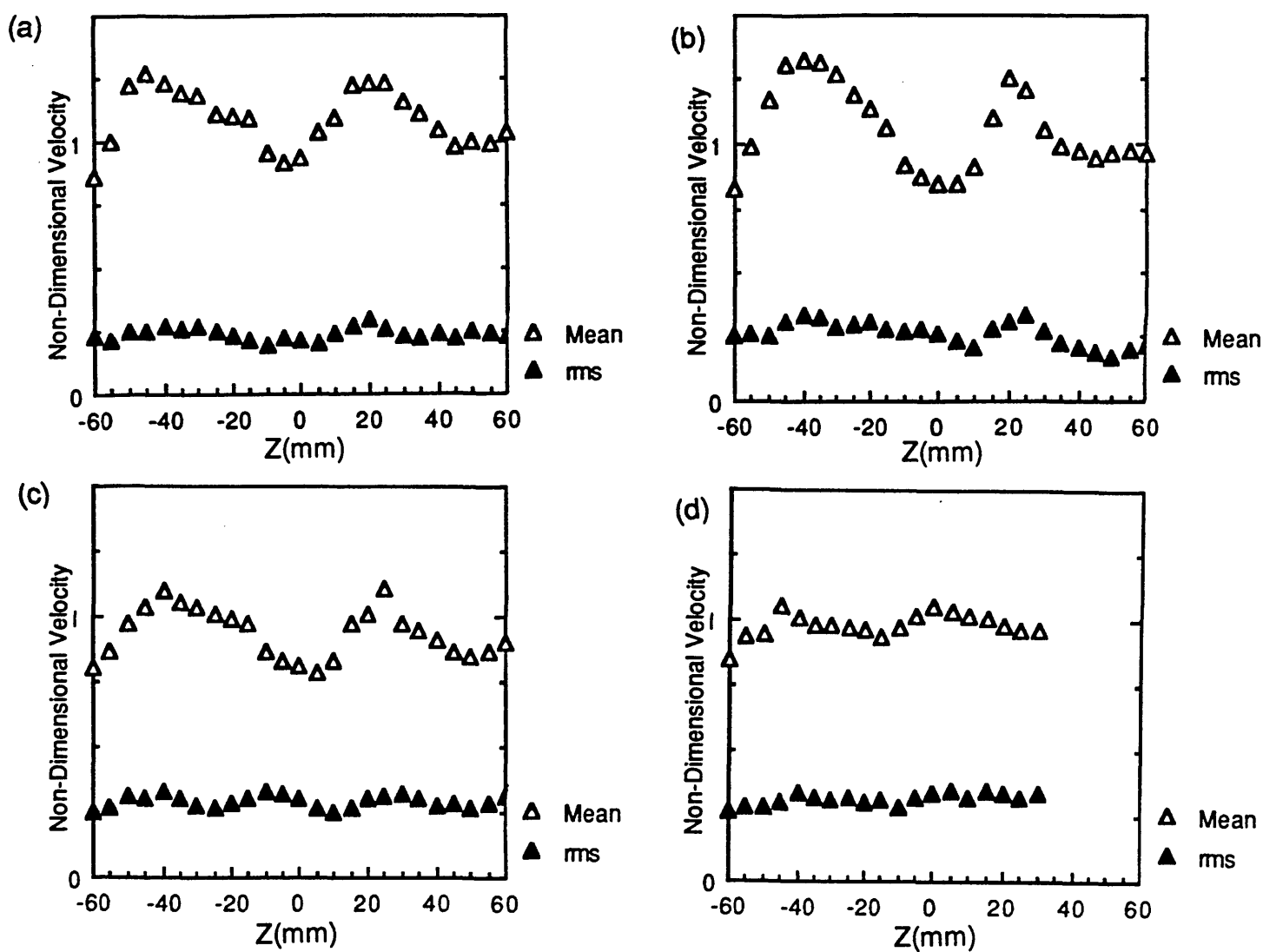


Figure 6.9 Profiles of Measured Mean and rms Streamwise Velocity at Entry to Curved Nozzle in Combusting Flow for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=60$
 (a) $y=5.5\text{mm}$ (b) $y=8\text{mm}$ (c) $y=13\text{mm}$
 (d) $y=18\text{mm}$

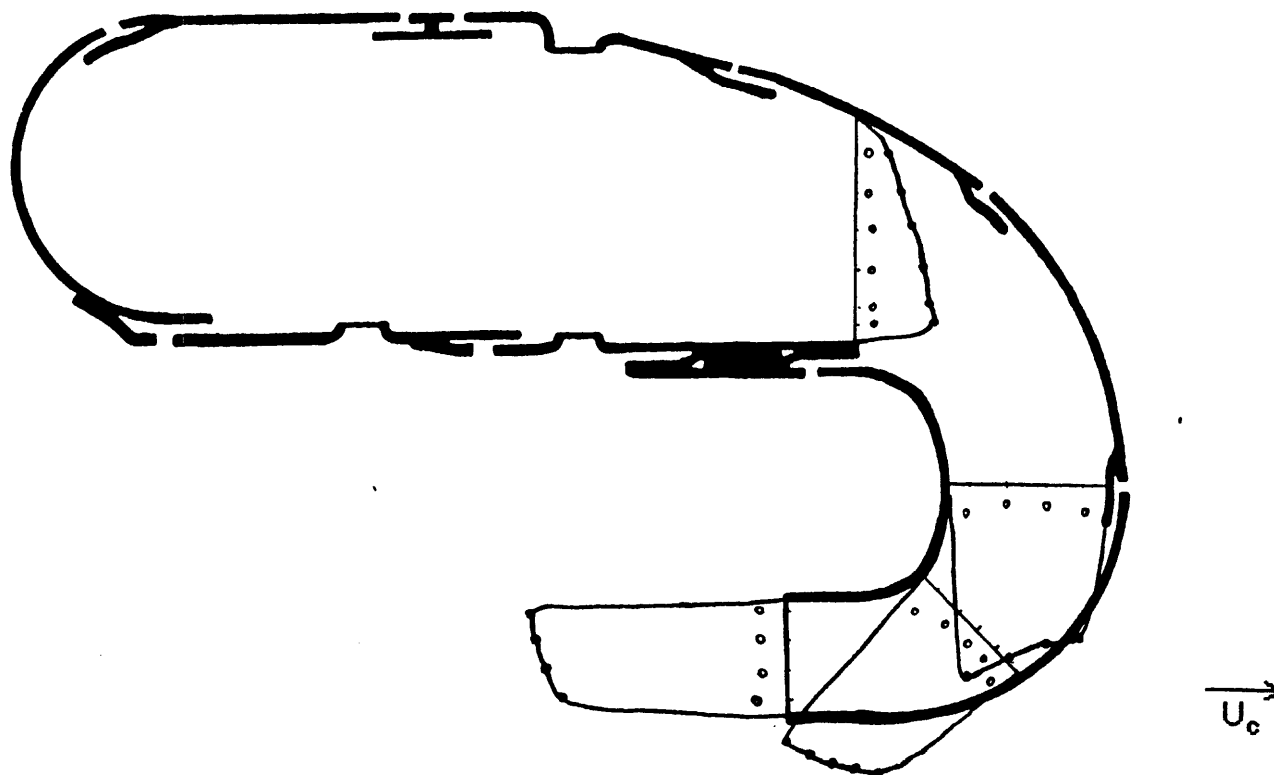


Figure 6.10 Profiles of Measured Mean and rms Streamwise Velocity at $z=0\text{mm}$ in Combusting Flow for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=60$

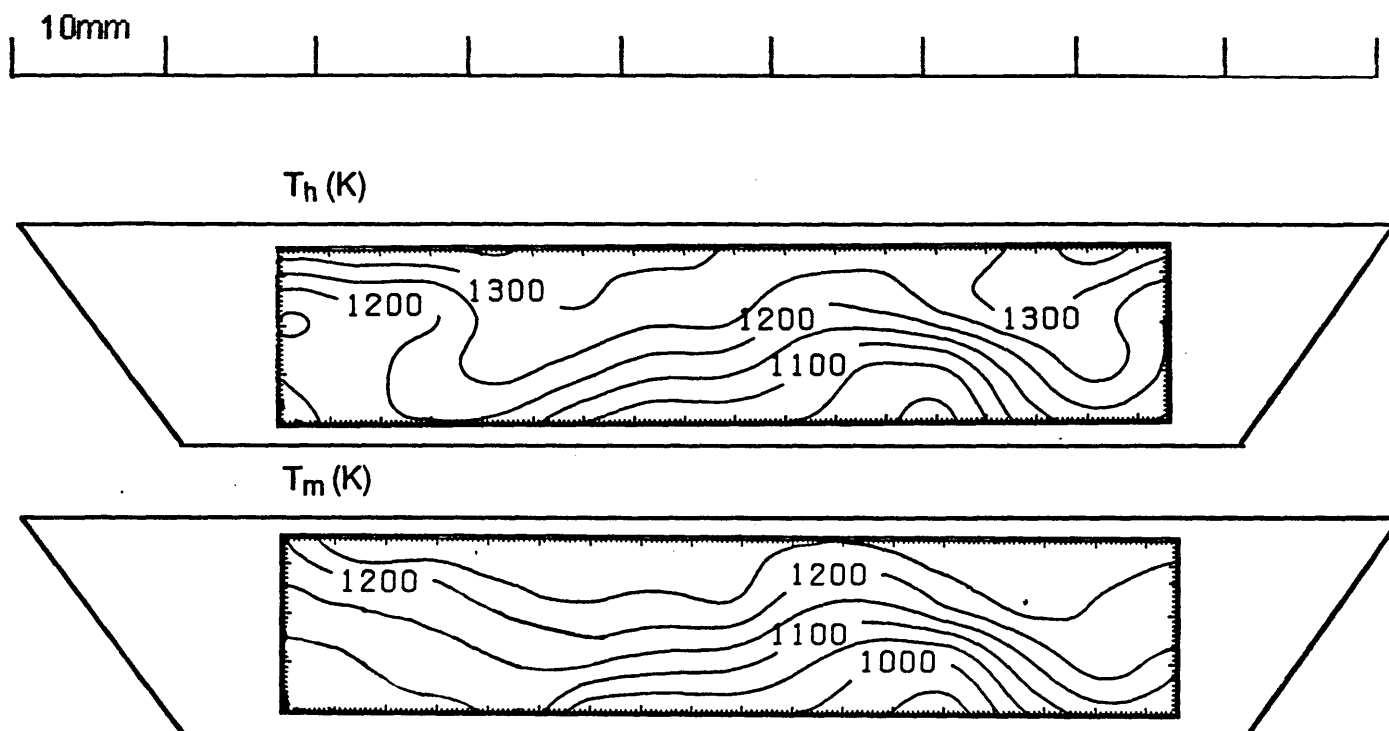


Figure 6.11 Measured and Enthalpy Balance Temperature in Exit Plane for $T_{inlet}=515K$ and $AFR=60$

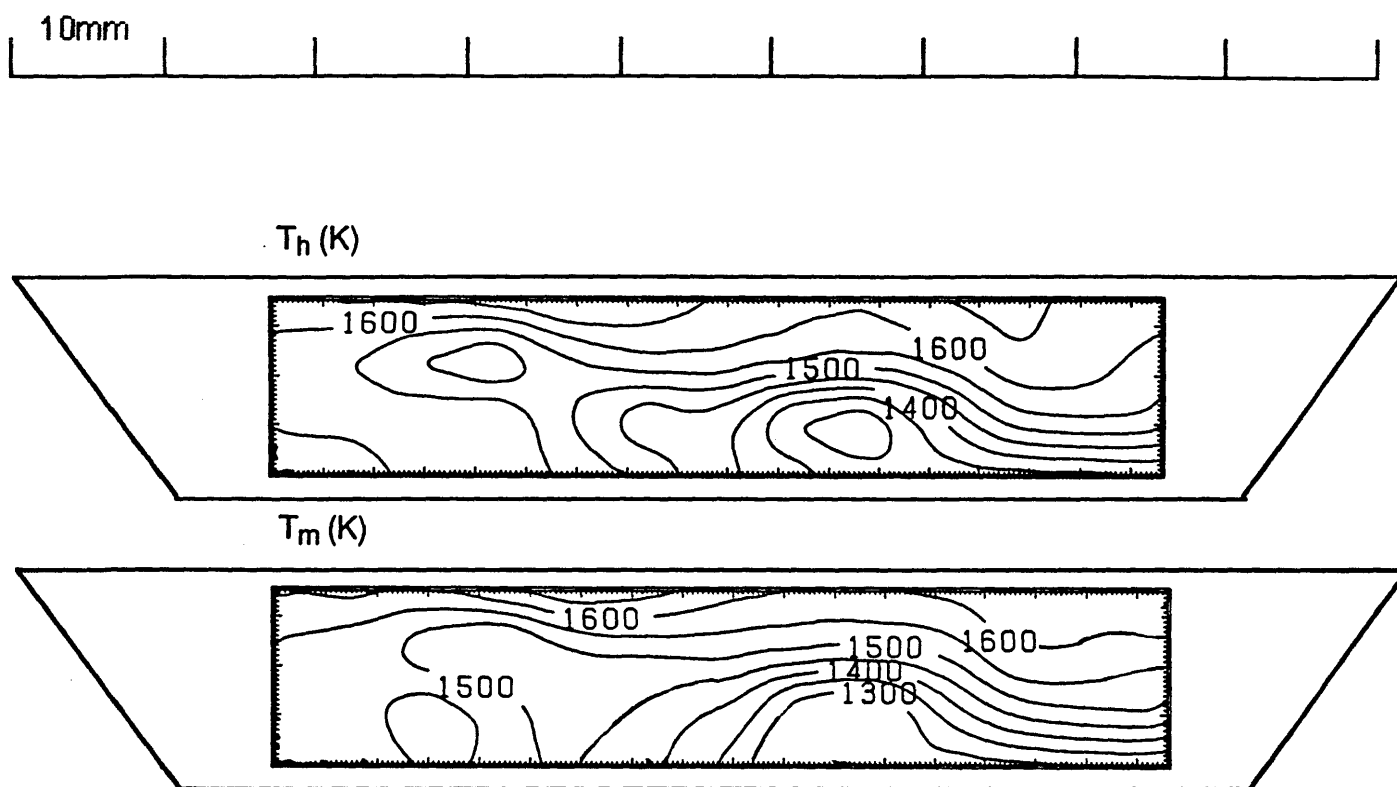


Figure 6.12 Measured and Enthalpy Balance Temperature in Exit Plane for $T_{inlet}=515K$ and $AFR=36$

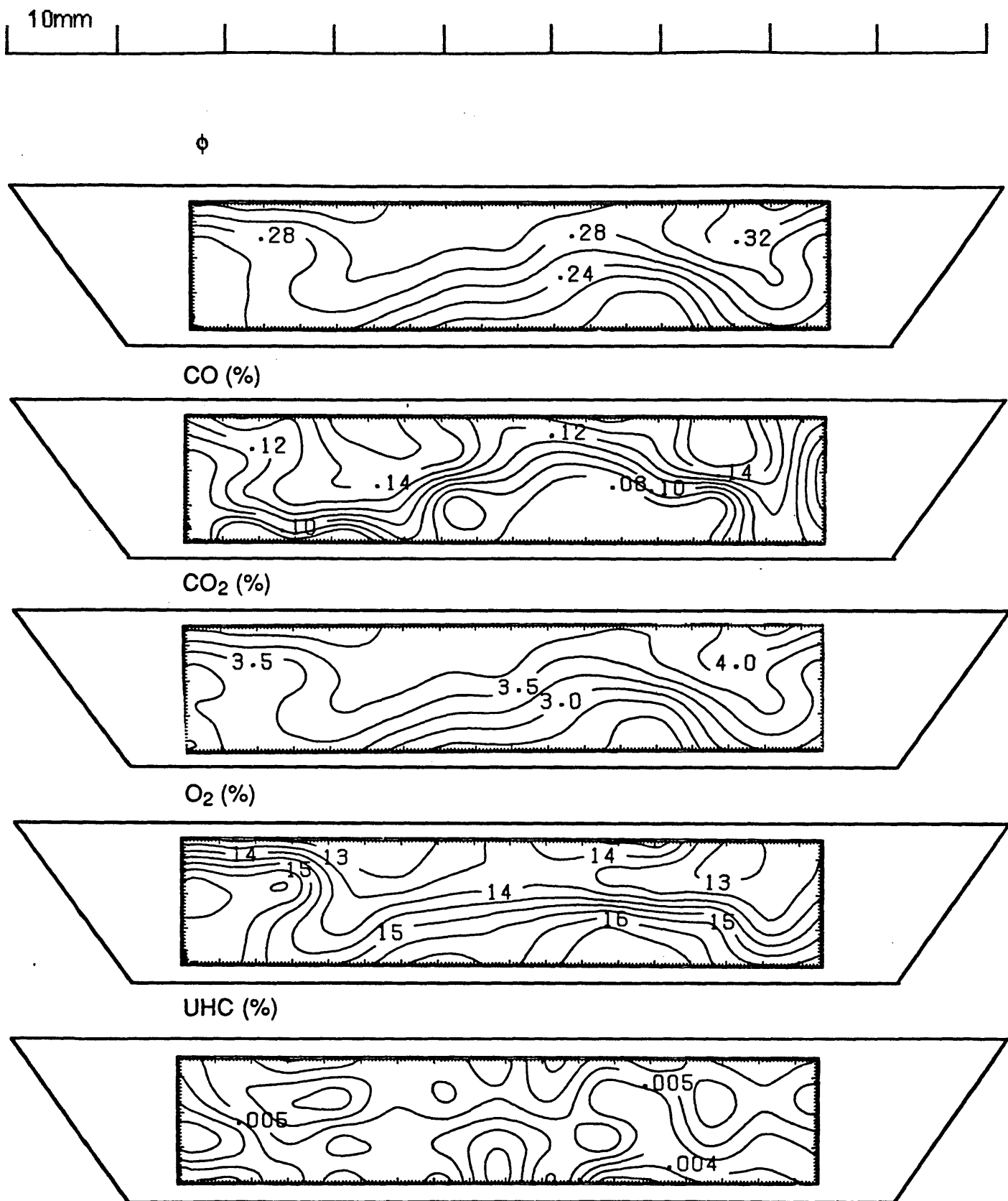


Figure 6.13 Distributions of Measured Species Concentrations in Exit Plane for
 $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=60$

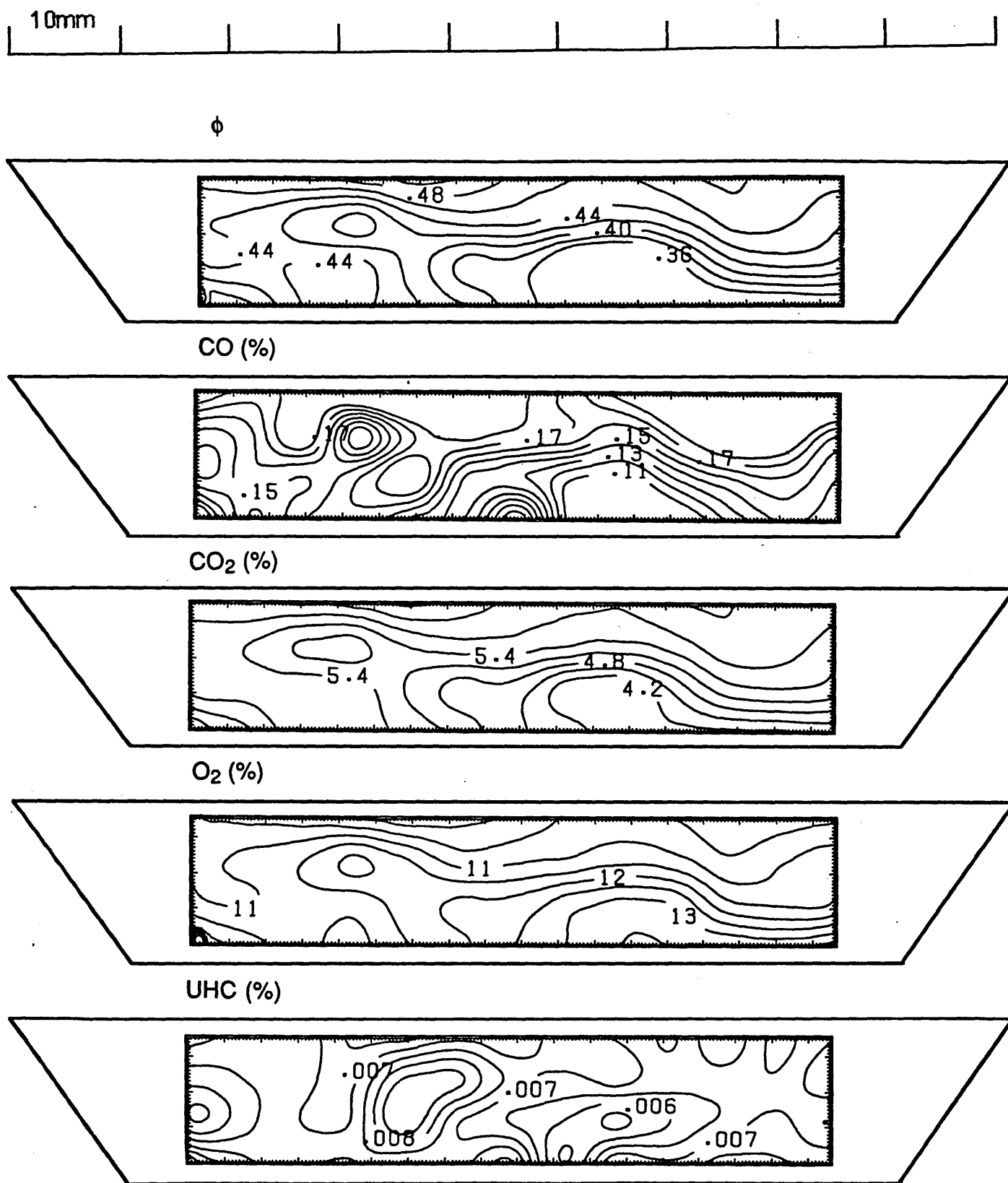


Figure 6.14 Distributions of Measured Species Concentrations in Exit Plane for $T_{\text{inlet}}=515\text{K}$ and $\text{AFR}=36$

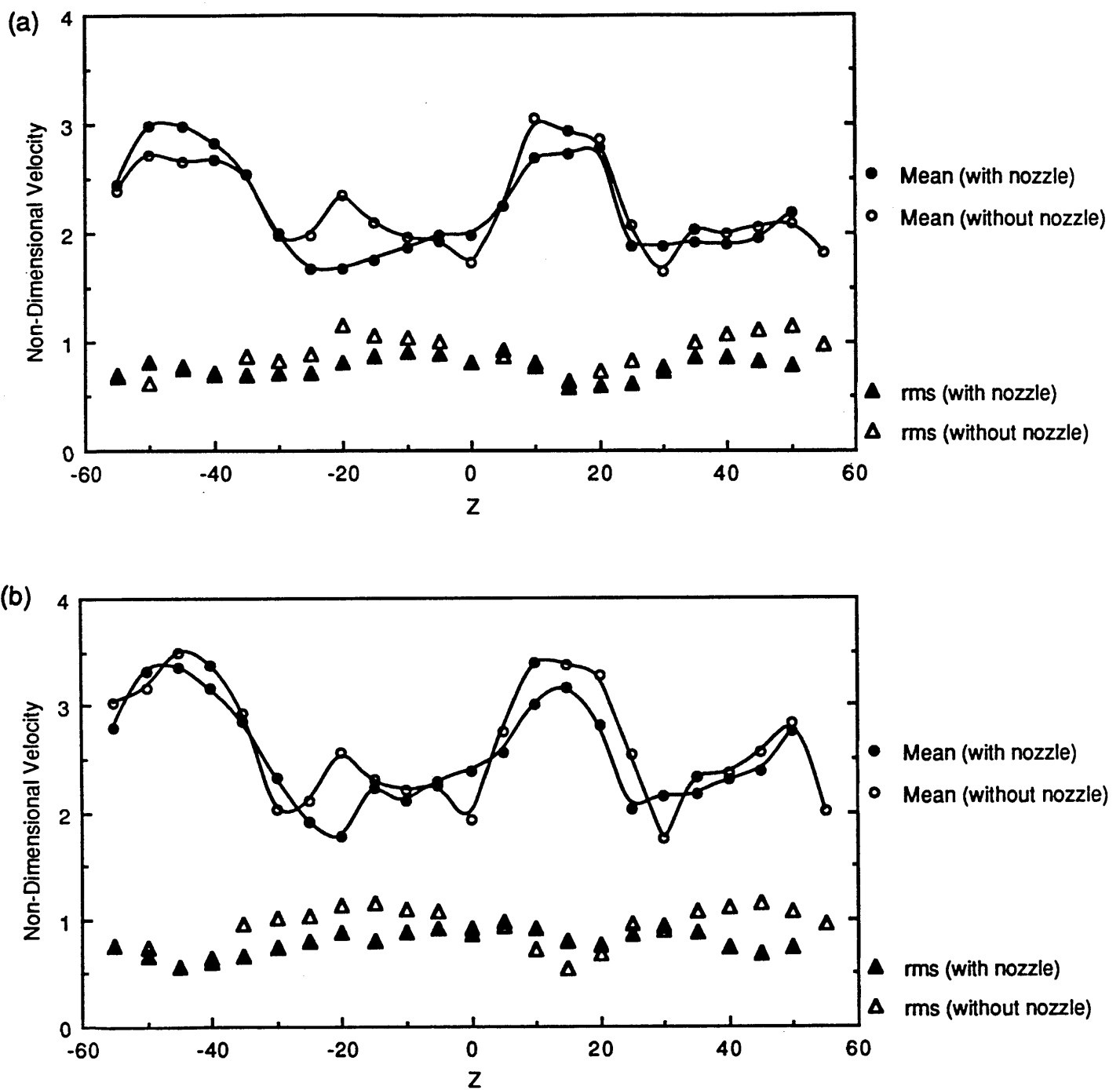
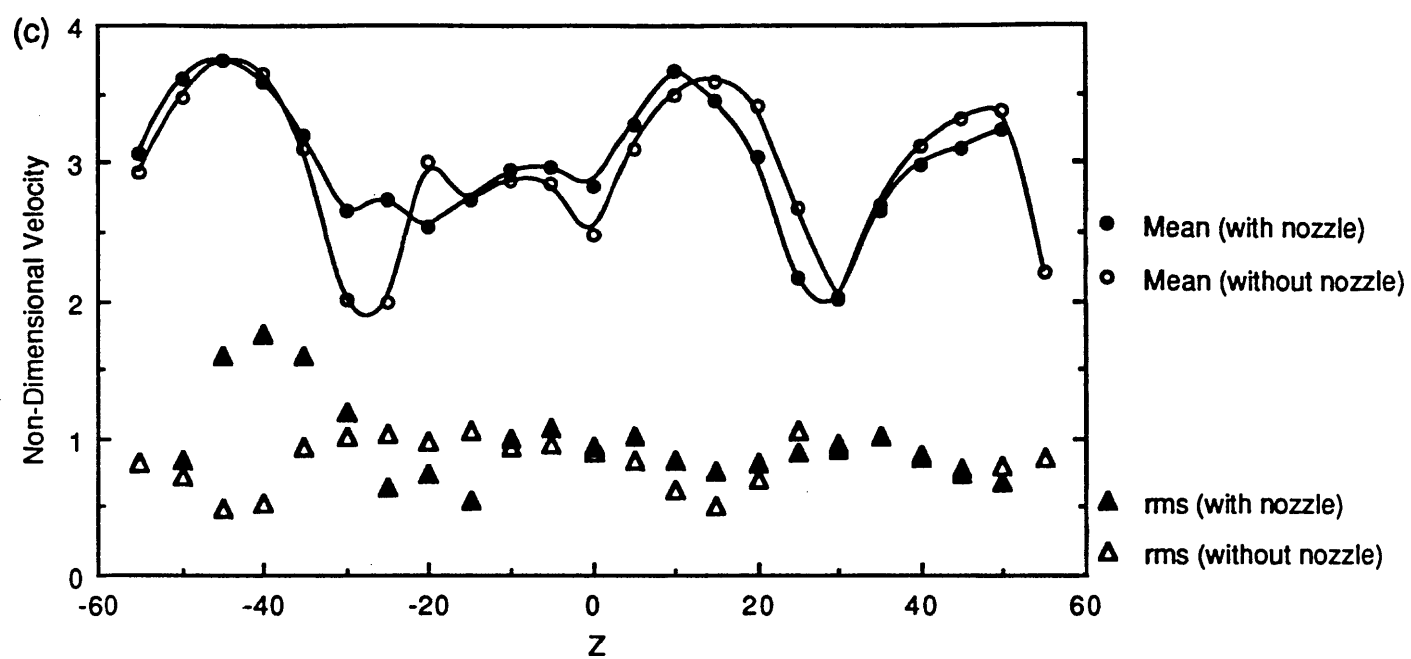


Figure 6.15 Comparison of Isothermal Streamwise Velocity With and Without the Curved Nozzle at $x=60\text{mm}$

(a) $y=0\text{mm}$ (b) $y=-5\text{mm}$ (c) $y=-10\text{mm}$



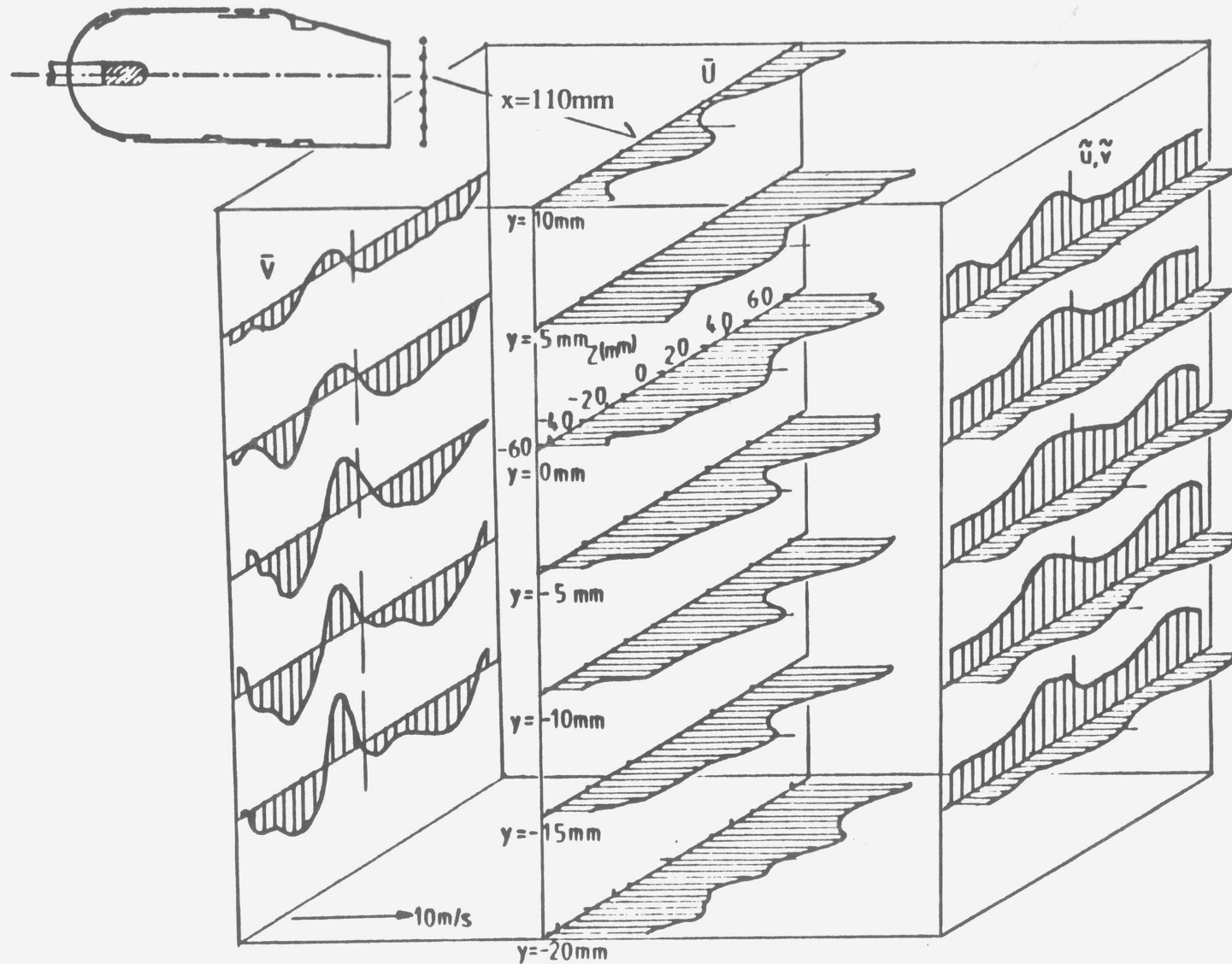


Figure 6.16 Profiles of Isothermal Streamwise Velocity Without Curved Nozzle at $x=110\text{mm}$ (Taken from Tse (1988))

CHAPTER 7 CLOSURE

Velocity and scalar characteristics inside models of annular combustor have been studied experimentally and computationally. This chapter summarises the major conclusions of the investigation, Section 7.1, and provides recommendations for further research, Section 7.2.

7.1 Conclusions

Chapters 4 to 6 present results obtained in a curved duct and two models of annular combustor and conclusions specific to those geometries are presented in the respective chapters. The following paragraphs bring together the major findings in the present study.

1. The flow development in the curved duct was independent of the inlet Reynolds number and, with a large aspect ratio, the effect of the secondary flow was limited in its extent in the spanwise direction to about one third of the width of the channel. Maximum streamwise velocities remained close to the inner wall due to the strong radial pressure gradient as a result of the sharp curvature. The calculations could predict the flow development in the core flow region where the influence of secondary flow was negligible but the flow characteristics in the near wall region could not be represented by the methodology adopted in the present study.
2. At take-off condition of the combustor with preheat air temperature of 515K, an over stoichiometric condition was observed in the primary zone and combustion continued beyond the dilution zone. CO to CO₂

conversion and oxidation of UHC took place within the curved exit nozzle which led to lower emission indices and higher combustion efficiency at the exit of the nozzle as compared to the entry.

3. At the ground-idle condition, the temperature at the inlet to the nozzle dropped below 1400K and the conversion of CO and fuel breakup ceased at this location. Therefore, the emission indices and combustion efficiency at the exit of the nozzle remained the same. Nevertheless, the pattern factor improved due to the strong contraction in cross-sectional area of the nozzle.

4. The enthalpy balance temperatures, calculated from the measured species concentrations, and the equilibrium concentrations of major species, derived from the measured equivalence ratios, were predicted to within 10% of the measured values in fuel-lean conditions. In fuel rich conditions, the breakdown of the hydrocarbon fuel was limited by finite-rate kinetics and therefore the equilibrium assumption was unable to give an accurate prediction of the local species concentrations.

5. Numerical experiments demonstrated that the flowfield inside combustor was sensitive to the inlet boundary conditions. This implies that measurements or empirical correlations of flow splits and jet angles are very important, or equivalently, combustor models must be extended to predict the flow in the external supply annulus.

7.2 Recommendations for Future Work

In continuing the investigation carried out in the present study, the following paragraphs suggest the areas for future work in order to improve understanding of the flow and combustion characteristics in the gas turbine combustor and enhance its performance in terms of combustion efficiency and reduction in pollution.

1. In a real gas turbine combustor, the inlet air temperature and pressure, typically about 700K and 14 bar, are considerably higher than those employed here. Higher preheat temperature would result in better combustion efficiency and exit plane pattern factor as observed in previous experiments and it is desirable to quantify the effect of preheat temperature with series of experiments including more realistic operating conditions. The combustion process has been demonstrated to be strongly pressure dependent in simple configurations. Hence, a pressurised combustor rig would be useful to reveal the actual chemical reaction inside the flame tube.
2. One of the limiting factors in designing the combustor is the highest wall temperature so that it is essential to quantify the distribution of wall temperature on the flame tube, through the use of thermal paint or embedded thermocouples.
3. In the present calculations, the effect of different flow splits and inlet conditions through air and fuel inlet ports were determined. A corresponding experimental study with a model combustor is desirable to compare the results and to provide a data base for design.

4. The calculations performed in the present study revealed the shortfall of the existing methodology to predict the flow inside gas turbine combustor. Advanced numerical methods, e.g. Reynolds-stress turbulence model and multi-scalar combustion model, have proved their advantages in some relatively simple flows. It is desirable to incorporate them in future numerical work to quantify their advantages in calculating gas turbine combustor flows.

REFERENCE

Aguirre-Saldivar, R. (1986), "Two scalar probability density function models for turbulent flames", Ph. D. thesis, University of London.

Altgeld, H., Jones, W. P. and Wilhelmi, J. (1983), "Velocity measurements in a confined swirl-driven recirculating flow", *Expt. in Fluids*, 1, 73.

Alvarez, J. (1990), "Jet in a cross-flow", Report of progress on work performed for Rolls-Royce plc and RAE/Pyestock.

Attya, A. M. and Whitelaw, J. H. (1981), "Velocity, temperature and species concentrations in unconfined kerosene spray flame", ASME Paper 81-WA/HT-47.

Attya, A. M. and Whitelaw J. H. (1984), "Measurements and calculations of preheated and unpreheated confined kerosene spray flame", *Comb. Sci. and Tech.*, 40, 193.

Azzola, J., Humphrey, J. A. C., Iacovides, H. and Launder, B. E. (1986), "Developing turbulent flow in a U-bend of circular cross-section: measurement and computation", *J. Fluid Engng.*, 108, 214.

Ballantyne, A. and Moss, J. B. (1977), "Fine-wire thermocouple measurements of fluctuation temperature", *Comb. Sci. and Tech.*, 17, 63.

Berger, S. A., Talbot, L. and Yao, L. S. (1983), "Flow in curved pipes", *Ann. Rev. Fluid Mech.*, 15, 461.

Bhangu, J. K., Snape, D. M. and Eardley, B. R. (1983), "The design and development of a low emissions transply combustor for the civil Spey engine", AGARD-CP353, Paper 23.

Bicen, A. F., Heitor, M. V. and Whitelaw, J. H. (1986), "Velocity and temperature measurements in a can-type gas-turbine combustor", AGARD CP-399, Paper 14.

Bicen, A. F. and Jones, W. P. (1986), "Velocity characteristics of isothermal and combusting flows in a model combustor", *Comb. Sci. and Tech.*, 49, 1.

Bicen, A. F., Senda, M. and Whitelaw, J. H. (1988), "Scalar characteristics of combusting flow in a model annular combustor", presented at the 33th ASME International Gas Turbine and Aeroengine Congress and Exposition, Amsterdam.

Bicen, A. F., Tse, D. G. N. and Whitelaw, J. H. (1987), "Flow characteristics of a model annular combustor", AGARD CP-422, Paper 14.

Bicen, A. F., Tse, D. G. N. and Whitelaw, J. H. (1988), "Flow and combustion characteristics of an annular combustor", *Comb. and Flame*, 71, 175.

Bicen, A. F., Tse, D. G. N. and Whitelaw, J. H. (1990), "Combustion characteristics of a model can-type combustor", *Comb. and Flame*, 80, 111.

Bilger, R. W. (1977), "Probe measurements in turbulent combustion", *AIAA Progress in Astronautics and Aeronautics*, 53, 49.

Bradley, D. and Matthews, K. I. (1968), "Measurements of high gas temperature with fine wire thermocouple", *J. Mech. Engng. Sci.*, 10, 299.

Brown, O. G. and Morris, A. W. (1963), "Turbulent flow of water in plane curved channels of finite depth", *J. Basic Engng.*, 85, 377.

Brum, R. D. and Samuelsen, G. S. (1987), "Two-component laser anemometry measurements of non-reacting and reacting complex flows in a swirl-stabilised model combustor", *Expt. in Fluids*, 5, 95.

Bryer, D. W. and Pankhurst, R. C. (1971), Pressure probe methods for determining wind speed and flow direction, HMSO.

Burrus, D. L., Shyy, W. and Braaten, M. E. (1987), "Numerical models for analytical predictions of combustor areothermal preformance characteristics", AGARD CP-422, Paper 25.

Cameron, C. D., Brouwer, J. and Samuelsen, G. S. (1988), "A model gas turbine combustor with wall jets and optical access for turbulent mixing, fuel effects, and spray studies", 22nd Symposium (Intl.) on Combustion, The Combustion Institute, 465.

Cameron, C. D., Brouwer, J., Wood, C. P. and Samuelsen, G. S. (1989), "A detailed characterization of the velocity and thermal fields in a model

can combustor with wall jet injection", *J. Engng. for Gas Turbines and Power*, 111, 31.

Chandran, S. B. S., Komerath, N. M., Grissom, W. M., Strahle, W. C. and Jagoda, J. I. (1984), "Time-resolved thermometry by simultaneous thermocouple and Rayleigh scattering measurements in a turbulent flame", *Comb. Sci. and Tech.*, 44, 47.

Chang, S. M., Humphrey, J. A. C. and Modavi, A. (1983), "Turbulent flow in a strongly curved U-bend and downstream tangent of square cross-section", *PhysicoChemical Hydrodynamics*, 4, 243.

Chigier, N. A. and Strokin, V. (1974), "Mixing processes in a free turbulent diffusion flame", *Comb. Sci. and Tech.*, 9, 111.

Choi, Y. D., Iacovides, H. and Launder, B. E. (1989), "Numerical computation of turbulent flow in a square-sectioned 180 degree bend", *J. Fluid Engng.*, 111, 59.

Chomiak, J. and Niedzialek, B. (1967), "Measurements of rapidly varying gas temperature in an unsteady flow", *Int. J. Heat and Mass Transfer*, 10, 1571.

Chow, S.K. and McGuirk, J.J. (1991), "Numerical Prediction of Flow and Combustion Characteristics of a Model Annular Combustor", presented at the 36th ASME International Gas Turbine and Aeroengine Congress and Exposition, Orlando.

Chow, S.K., Senda, M. and Whitelaw, J.H. (1989), "Combustion Characteristics of a Liquid-Fuelled Model Annular Combustor", presented at the 9th International Symposium on Air Breathing Engines, Athens.

Chow, S.K., Tse, D.G.N. and Whitelaw, J.H. (1990), "Review of Recent Measurements in Gas Turbine Combustor", Proceedings of NATO Advanced Study Institute on Instrumentation for Combustion in Engine.

Chow, S.K. and Whitelaw, J.H. (1991), "Scalar Characteristics in a Liquid-Fuelled Combustor with a Curved Exit Nozzle", presented at the IUTAM Symposium on Aerothermodynamics in Combustors, Taipei.

Colket, M. B., Chiappeta, L., Guile, R. N., Zabielski, M. F. and Seery, D. J. (1982), "Internal aerodynamics of gas sampling probes", *Comb. and Flame*, 44, 3.

Correa, S. M. (1985), "A model for non-premixed turbulent combustion of CO/H₂ jets", presented at the 9th International Symposium on Combustion Processes, Poland.

Coupland, J. and Priddin, C. H. (1987), "Modelling the flow and combustion in a production gas turbine combustor", *Turbulent Shear Flow* 5, (edited by Bradbury et al), Springer Verlag.

Drain, L. E. (1980), The laser Doppler technique, John Wiley & Sons.

Durst, F., Melling, A. and Whitelaw, J. H. (1981), Principles and practice of laser Doppler anemometry, (2nd edition), Academic Press.

El-Banhawy, Y. and Whitelaw, J. H. (1980), "Calculation of the flow properties of a confined kerosene-spray flame", *AIAA Journal*, 18, 1503.

Enayet, M. M., Gibson, M. M., Taylor, A. M. K. P. and Yianneskis, M. (1981), "Measurements of laminar and turbulent flow in curved ducts", Report FS/81/7, Mech. Eng. Dept., Imperial College.

Gibson, M. M., Jones, W. P., McGuirk, J. J. and Whitelaw, J. H. (1984), "Post experience course in turbulence models for computational fluid dynamics", Lecture notes, Mech. Eng. Dept., Imperial College.

Glass, M. and Bilger, R. W. (1978), "The turbulent jet diffusion flame in a co-flowing stream - some velocity measurements", *Comb. Sci. and Tech.*, 18, 165.

Godoy, S. (1982), "Turbulent diffusion flames", Ph. D. Thesis, University of London.

Gordon, S. and McBride, B. J. (1971), "Computer program for calculation of complex chemical equilibrium compositions", NASA SP273.

Gosman, A. D. and Ioannides, E. (1983), "Aspects of computer simulation of liquid-fueled combustors", *J. Energy*, 6, 482.

Green, A. and Whitelaw, J. H. (1980), "Measurements and calculations of the isothermal flow in axisymmetric models of combustor geometries", *J. Mech. Engng. Sci.*, 22, 119.

Gunston, B. (1989), Rolls-Royce aero-engine, Patrick Stephens Limited.

Heitor, M. V. (1985), "Experiments in turbulent reacting flow", Ph. D. Thesis, University of London.

Heitor, M. V. and Whitelaw, J. H. (1986), "Velocity, temperature and species characteristics of the flow in a gas-turbine combustor", *Comb. and Flame*, 64, 1.

Hille, P., Vehrenkamp, R. and Schulz-Dubios, E. O. (1985), "The development and structure of primary and secondary flow in a curved square duct", *J. Fluid Mech.*, 151, 219.

Holderness, F. H., Tilson, J. R. and MacFarlane, J. J. (1969), "Electrical compensation for radiation loss in thermocouples", NGTE Note No. 758.

Humphrey, J. A. C., Iacovides, H. and Launder, B. E. (1985), "Some numerical experiments on developing laminar flow in circular-sectioned bends", *J. Fluid Mech.*, 154, 357.

Humphrey, J. A. C., Taylor, A. M. K. P. and Whitelaw, J. H. (1977), "Laminar flow in a square duct of strong curvature", *J. Fluid Mech.*, 83, 509.

Humphrey, J. A. C., Whitelaw, J. H. and Yee, G. (1981), "Turbulent flow in a square duct with strong curvature", *J. Fluid Mech.*, 103, 443.

Janjua, S. I., McLaughlin, D. K., Jackson, T. W. and Lilley, D. G. (1983), "Turbulence measurements in confined jets using a rotating single-wire techniques", *AIAA Journal*, 21, 1609.

Johnston, J. P. (1978), "Internal flow", in *Turbulence*, (edited by P. Brawshaw) *Topics in Applied Physics*, 12.

Jones, W. P. (1971), "Laminarisation in strongly accelerated boundary layer", Ph. D. Thesis, University of London.

Jones, W. P. (1980), "Models for turbulent flows with variable density", VKI Lecture Series 1979-2.

Jones, W. P. and Launder, B. E. (1972), "The prediction of laminarisation with a two-equation model of turbulence", *Int. J. Heat and Mass Transfer*, 15.

Jones, W. P. and Launder, B. E. (1973), "Prediction of low Reynolds number phenomena with a two-equation model of turbulence", *Int. J. Heat and Mass Transfer*, 16, 1189.

Jones, W. P. and Linstedt, R. P. (1988a), "Global reaction schemes for hydrocarbon combustion", *Comb. and Flame*, 73, 233.

Jones, W. P. and Linstedt, R. P. (1988b), "The calculation of the structure of laminar counterflow diffusion flame using a global reaction mechanism", *Comb. Sci. and Tech.*, 61, 31.

Jones, W. P. and McGuirk, J. J. (1979), "Mathematical modelling of gas-turbine combustion chambers", AGARD CP-275, Paper 4.

Jones, W. P., McGuirk, J. J. and Sodha, M. N. (1989), "Calculation of the flow in a sector of an annular combustor", *J. Power Engng.*, 203, 187.

Jones, W. P. and Toral, H. (1983), "Temperature and composition measurements in a research gas turbine combustor chamber", *Comb. Sci. and Tech.*, 31, 249.

Jones, W. P. and Whitelaw, J. H. (1982), "Calculation methods for reacting turbulent flows: a review", *Comb. and Flame*, 48, 1.

Jones, W. P. and Whitelaw, J. H. (1984), "Modelling and measurements in turbulent combustion", 20th Symposium (Intl.) on Combustion, The Combustion Institute, 233.

Kennedy, I. M. and Kent, J. H. (1981), "Scalar measurements in a co-flowing turbulent diffusion flame", *Comb. Sci. and Tech.*, 25, 109.

Kita, Y., Okazaki, M., Hirose, K. and Tanaka, S. (1987), "Fully developed turbulent flow in a curved rectangular duct", *JSME International Journal*, 30, 1558.

Koutmos, P. (1985), "An isothermal study of gas turbine combustor flows", Ph. D. thesis, University of London.

Kramlich, J. C. and Malte, P. C. (1980), "Modelling and measurements of a sampling probe effects on pollutant gases drawn from flame zone", *Comb. Sci. and Tech.*, 18, 91.

LaRue, J. C., Samuelsen, G. S. and Seiler, E. T. (1984), "Momentum and heat-flux in a swirl stabilised combustor", 20th Symposium (Intl.) on Combustion, The Combustion Institute, 277.

Lauder, B. E. and Spalding, D. B. (1974), "The numerical computation of turbulent flow", *Computer methods in App. Mech. and Engng.*, 3, 269.

Lefebvre, A. H. (1983), Gas turbine combustion, Hemisphere Publishing Corporation.

Lefebvre, A. H. (1990), Atomisation and sprays, Hemisphere Publishing Corporation.

Lenz, W. and Gunther, R. (1980), "Measurements of fluctuating temperature in a free-jet diffusion flame", *Comb. and Flame*, 37, 63.

Leonard, B. P., Lenchziner, M. A. and McGuirk, J. J. (1978), "Third order finite-difference method for steady two-dimensional convection", *Proc. 1st Int.Conference on numerical methods in laminar and turbulent flows*, (edited by Taylor), Pentech. Press.

Levy, R., Briley, W. R. and McDonald, H. (1983), "Viscous primary/secondary flow analysis for use with non-orthogonal co-ordinate systems", AIAA 21st Aerospace Sciences Meeting.

Lockwood, F. C. and Moneib, (1981), "Fluctuation temperature measurements in a heated round free jet", *Comb. Sci. and Tech.*, 22, 63.

Lorenzetto, G. E. and Lefebvre, A. H. (1977), "Measurements of drop size on a plain jet airblast atomiser", *AIAA Journal*, 15, 1006.

Magre, P., Moreau, P., Collin, G., Borghi, R. and Pealat, M. (1988), "Further studies by CARS of premixed turbulent combustion in a high velocity flow", *Comb. and Flame*, 71, 147.

Malte, P. C. and Kramlich, J. C. (1980), "Further observations of the effect of sample probes on pollutant gases drawn from flame zones", *Comb. Sci. and Tech.*, 22, 263.

Manners, A. P. (1988), "The calculation of the flows in gas-turbine combustor system", Ph. D. Thesis, University of London.

Masri, A. R. and Bilger, R. W. (1988), "Turbulent non-premixed flames of methane near extinction: mean structure from Raman measurements", *Comb. and Flame*, 71, 245.

McGuirk, J. J. (1987), "3D combustor prediction - a comparison of equilibrium and laminar flamelet chemistry models", AGARD CP-422, Paper 31.

Mellor, A. M. (1972), "Current kinetic modelling techniques for continuous flow combustors", Proceeding of Symposium on Emissions from Continuous Combustion System, Plenum Press.

Mellor, A. M. (1976), "Gas turbine engine pollution", *Prog. Energy Comb. Sci.*, 1, 111.

Mellor, A. M. (1990), (eds.), Design of modern turbine combustor, Academic Press.

Milosavljevic, V. (1989), private communication.

Mongia, H. C. (1987), "A status report on gas turbine combustor modeling", AGARD CP-422, Paper 26.

Nicholls, J. A., Kauffman, C. W., Pelaccio, D. G., Glass, D. R. and Driscoll, J. F. (1980), "The effect of fuel sprays on emission from a research gas turbine combustor", *Comb. Sci. and Tech.*, 23, 203.

Noyce, J. R., Sheppard, C. G. W. and Yamba F. D. (1981), "Measurements of mixing and species concentrations within a gas turbine type combustor", *Comb. Sci. and Tech.*, 25, 209.

Noyce, J. R. and Sheppard, C. G. W. (1982), "The influence of equivalence ratio variation on pollutant formation in a gas turbine type combustor", *Comb. Sci. and Tech.*, 29, 37.

Nukiyama, S. and Tanasawa, Y. (1939), "Experiments on atomisation of liquids in an air stream", *Trans. Soc. Mech. Engineers (Japan)*, 5, 68.

Onuma, Y. and Ogasawara, M. (1974), "Studies on the structure of a spray combustion flame", 15th Symposium (Intl.) on Combustion, The Combustion Institute, 453.

Palma, J. M. J. M. (1988), "Mixing in non-reacting gas-turbine combustor system", Ph. D. Thesis, University of London.

Pascau, A. (1988), "Predictions of strongly swirling flows with second order turbulence models", Report of progress on work performed for Rolls-Royce plc and RAE/Pystock.

Patankar, S. V. (1980), Numerical heat transfer and fluid flow, Hemisphere Publishing Coporation.

Patankar, S. V. and Spalding, D. B. (1972), "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows", *Int. J. Heat and Mass Transfer*, 15, 1787.

Peters, N. (1986), "Laminar flamelet concepts in turbulent combustion", 21st Symposium (Intl.) on Combustion, The Combustion Institute, 1231.

Pratap, V. S. (1975), "Flow and heat transfer in curved duct", Ph. D. Thesis, University of London.

Rhie, C. M. and Chow, W. L. (1983), "Numerical study of the turbulent flow past an airfoil with trailing edge separation", *AIAA Journal*, 21, 1525.

Rhode, D. L., Lilley, D. G. and McLaughlin, D. K. (1983), "Mean flowfields in axisymmetric combustor geometries with swirl", *AIAA Journal*, 21, 593.

Rizk, N. K. and Mongia, H. C. (1986), "Gas turbine combustor design methodology", AIAA Paper 86-1531.

Rizkalla, A. A. and Lefebvre, A. H. (1975), "The influence of air and liquid properties on airblast atomiser", *J. Fluid Engng.*, 97, 316.

Roache, P. J. (1976), Computational fluid dynamics, Hermosa Publishers.

Rodi, W. (1984), "Turbulence models and their application in hydraulics - a state of the art review", 2nd edition, Delft: International Association for Hydraulic Research.

Rojas, J., Whitelaw, J. H. and Yianneskis, M. (1983), "Flow in sigmoid diffusers of moderate curvature", Report FS/83/21, Mech. Eng. Dept., Imperial College.

Rowe, M. (1970), "Measurements and computations of flow in pipe bends", *J. Fluid Mech.*, 43, 771.

Schlichting, H. (1968), Boundary layer theory, McGraw Hill.

Schoenung, S. M. and Hanson, R. K. (1981), "CO and temperature measurements in a flat flame by laser absorption spectroscopy and probe techniques", *Comb. Sci. and Tech.*, 24, 227.

Serag-Eldin, M. A. and Spalding, D. B. (1979), "Computations of three-dimensional gas-turbine combustion chamber flows", *J. Engng for Power*, 101, 327.

Shearer, A. J., Tamura, H. and Faeth, G. M. (1979), "Evaluation of a locally homogeneous flow model of spray evaporation", *J. Energy*, 3, 271.

Shelef, M., Otto, K. and Gandhi, H. (1968), "The oxidation of CO by O₂ and by NO on supported chromium oxide and other oxide metal catalyst", *J. Catalyst*, 12, 361.

Shisler, R. A., Tuttle, J. H. and Mellor, A. M. (1975), "Emission from and within a film-cooled combustor", *Comb. Sci. and Tech.*, 11, 153.

Shyy, W., Braaten, M. E. and Burrus, D. L. (1989), "Study of three-dimensional gas-turbine combustor flows", *Int. J. Heat and Mass Transfer*, 32, 1155.

So, R. M. C., Whitelaw, J. H. and Mongia, H. C. (1988), "Introductory Remarks", *Comb. Sci. and Tech.*, 58, 1.

Sodha, M. N. (1988), "Predictions with irregular geometry and castellated PACE", Report of progress on work performed for Rolls-Royce plc and RAE/Pyestock.

Sotharan, A. (1984), "The Rolls-Royce annular vaporiser combustor", *J. Engng. for Gas Turbines and Power*, 106, 88.

Squire, H. B. and Winter, K. G. (1951), "The secondary flow in a cascade of airfoils in a non-uniform stream", *J. Aeronaut. Sci.*, 18, 271.

Taylor, A. M. K. P., Whitelaw, J. H. and Yianneskis, M. (1981), "Measurements of laminar and turbulent flow in a curved duct with thin inlet boundary layers, NASA contractor report 3367.

Tennekes, H. and Lumley, J. L. (1972), A first course in turbulence, MIT Press.

Thompson, J. F. (1980), Computational fluid dynamics, Hemisphere Publishing Corporation.

Tober, A. (1990), "Experiments in kerosene-fuelled model TAY combustor", Report of progress on work performed for Rolls-Royce plc and RAE/Pyestock.

Toral, H. and Whitelaw, J. H. (1982), "Velocity and scalar characteristics of the isothermal and combusting flow in a combustor sector rig", *Comb. and Flame*, 45, 251.

Tse, D. G. N. (1988), "Flow and Combustion Characteristics of Annular and Can-Type Combustor", Ph. D. thesis, University of London.

Tuttle, J. H., Shisler, R. A. and Mellor, A. M. (1976), "Investigation of liquid fuelled turbulent diffusion flames", *Comb. Sci. and Tech.*, 14, 229.

Vranos, A. and Taback, E. F. (1976), "Combustion product distribution in the primary zone of a gas turbine combustor", *Comb. and Flame*, 46, 129.

Wilhelmi, J. (1984), "Axisymmetric swirl stabilised combustor", Ph. D. Thesis, University of London.

Yanta, W. J. and Smith, R. A. (1978), "Measurements of turbulent-transport properties with a laser Doppler velocimeter", AIAA paper 73-169.

Zhang, J. Q. and Cheng, J. S. (1986), "Determination of the temperature profile of axisymmetric combustion-gas flow from infrared spectral measurements", *Comb. and Flame*, 65, 163.