

Cranfield Institute of Technology

Riaz Sanatian

Computational and Experimental Models of Induction Flows in Spark-Ignition Engines.

School of Mechanical Engineering

Ph.D. Thesis

C R A N F I E L D I N S T I T U T E O F T E C H N O L O G Y

School of Mechanical Engineering

Ph.D. Thesis

R. SANATIAN

Computational and Experimental Models of Induction Flows
in Spark-Ignition Engines

Supervisor: J.B. Moss

March 1988

To Malihe

ACKNOWLEDGEMENTS

I would like to offer my most sincere and heart-felt thanks to my supervisor, Professor J.B. Moss; for his interest, guidance and support in carrying out this work. Thanks, also, to Mr. D. Binks for his help in the flow visualization studies.

Finally, thanks to my dear wife, Malihe, for the task of typing the text.

ABSTRACT

The objective of this thesis is to combine computational flow modelling, flow visualization and point measurements of mean flow and turbulence properties to obtain a better, more detailed, understanding of the effects of alternative throttling devices on mixture preparation and turbulence generation in spark ignition engines. In so doing, it also seeks to assess the wider diagnostic potential of flow field computational techniques in internal combustion engine designs.

Full-scale models, comprising simplified representations of the induction tract, throttling device, inlet valve and cylinder, have been manufactured in Perspex for steady-state water analogy tests. The resulting photographs of flow tracers in a variety of viewing planes provide a clear, but qualitative, picture of the principal features of the flow in the models under study.

The essentially qualitative data obtained from water analogy tests are complemented by limited hot wire velocity measurements at particular stations in the Perspex models, with air replacing the water as the flow medium.

These data, supplemented by information in the literature, provide the framework for comparisons with an extensive computational simulation of induction flows which are performed using the general purpose PHOENICS code developed by CHAM. These studies include both transient and steady state predictions. The statistically stationary turbulent flow field through alternative induction system throttling devices -a conventional butterfly valve and a variable geometry ramp restriction- are modelled computationally and compared with water analogy flow visualization. The principal flow field characteristics are satisfactorily reproduced, including in particular the extent of the recirculation zone in the lee of the throttle and the relative persistence of the turbulence generated downstream for varying throat apertures. That generated by the two-dimensional variable geometry ramp is predicted to be both higher and persist beyond the inlet valve into the cylinder producing discernible swirl at high throttle settings. The limited quantitative comparisons with hot wire velocity measurements lend further support to the more detailed aspects of the computational predictions.

Finally, comparisons are made between PHOENICS predictions and Laser-Doppler measurements of velocity for transient flow inside an axisymmetric motored piston-cylinder assembly, for different valve seat angles, reported in the literature. The agreement is again very encouraging, reinforcing the view that general purpose computer codes of the kind investigated can play an important role in detailed design assessment and evaluation.

CONTENTS

ACKNOWLEDGEMENTS	I
ABSTRACT	II
CONTENTS	III
NOMENCLATURE	VIII
CHAPTER 1: INTRODUCTION	1
CHAPTER 2: TURBULENT COMBUSTION IN SPARK-IGNITION ENGINES	
2.1 Introduction	4
2.2 Turbulent flow in spark-ignition engines	4
2.2.1 General features of the turbulent flow field	4
2.2.2 Intake process	6
2.2.3 Compression process	7
2.3 Effects of engine variables on turbulence in spark-ignition engines	9
2.3.1 Engine speed	9
2.3.2 Engine compression ratio	11
2.3.3 Engine volumetric efficiency	11
2.4 Turbulent combustion and flame propagation in the spark-ignition engine	12
2.4.1 Turbulent flame propagation	12
2.4.2 Turbulent combustion process in spark-ignition engines	13
2.4.3 The influence of turbulence on ignition	14
2.4.4 Influence of turbulence on ignition delay	15
2.4.5 Influence of turbulence on flame propagation in engines	16
2.5 Concluding remarks	16

CHAPTER 3: REVIEW OF PREVIOUS STUDIES ON LEAN MIXTURE COMBUSTION

3.1	Introduction	18
3.2	Generation of motion in the combustion chamber of a spark-ignition engine and its effect on combustion	20
3.2.1	Squish	20
3.2.2	Swirl	22
3.3	Mixture motion generated in the induction manifold	24
3.3.1	Fuel-air mixing and distribution prior to the combustion chamber	24
3.3.2	Mixture quality	24
3.3.3	Generation of mixture turbulence through throttling devices and vortex generators	26
3.4	The variable ramp throttling device	29
3.4.1	Description of design	29
3.4.2	Principle of operation	30
3.4.3	Tests and results	30
3.4.4	Computer simulation studies	32
3.5	Investigative methods and approaches	33
3.6	Concluding remarks	36

CHAPTER 4: AERODYNAMIC FLOW VISUALIZATION STUDIES OF INDUCTION FLOWS

4.1	Introduction	38
4.2	Choice of working fluid and similarity conditions	40
4.2.1	Choice of working fluid	40
4.2.2	Flow visualization studies with water	42
4.2.3	Dynamic similarity conditions	44
4.3	Equipment and set up	46
4.3.1	Transparent models of the induction and throttling systems	46
4.3.2	Lighting and photographic equipment for flat beam technique	49
4.3.3	Arrangement of the water model network	51
4.4	Experimental procedure	53
4.4.1	Throttle setting	53

4.4.2	Flow field visualization test procedures for the variable geometry ramp throttling model	54
4.4.3	Flow visualization tests for the butterfly valve throttling model	57
4.5	Discussion of results	58
4.5.1	Variable geometry ramp results	58
4.5.1.1	Ramp angle 110°	58
4.5.1.2	Ramp angle 130°	60
4.5.1.3	Ramp angle 150°	60
4.5.1.4	Ramp angle 180°	61
4.5.2	Butterfly valve results	61
4.5.2.1	Butterfly valve angle 30°	61
4.5.2.2	Butterfly valve angle 50°	62
4.5.2.3	Butterfly valve angle 65°	62
4.5.2.4	Butterfly valve angle 90°	62
4.5.3	Comparison of the results of the two alternative systems	63
4.6	Concluding remarks	63
	Flow visualization photographs	65

CHAPTER 5: STEADY FLOW VELOCITY MEASUREMENTS OF THE INDUCTION FLOW

5.1	Introduction	81
5.2	Measurement techniques	81
5.2.1	Physical characteristics of hot wires	82
5.2.2	Principles of hot wire operation	83
5.2.3	Constant temperature anemometer	85
5.3	Equipment and set up	87
5.3.1	Instrumentation for hot wire velocity measurements	87
5.3.2	Testing set up	89
5.4	Experimental procedure	90
5.4.1	Hot wire calibration	90
5.4.2	Steady flow tests	92
5.4.3	Choice of flow rates tested	94
5.4.4	Hot wire orientation and interpretation of its measurements	96

5.5	Discussion of results	100
	Flow measurement figures	105
CHAPTER 6: COMPUTER MODELS OF THE INDUCTION FLOW IN ALTERNATIVE THROTTLING SYSTEMS		
6.1	Introduction	143
6.2	Computational fluid mechanics	144
6.2.1	Mathematical description of physical phenomena	145
6.2.2	Nature of numerical methods	146
6.3	PHOENICS computer code	147
6.3.1	The structural principle of PHOENICS	148
6.3.2	Mathematical basis of PHOENICS	149
6.3.3	The solution procedure	150
6.3.4	Porosity specification	152
6.3.5	Incorporation of boundary conditions	153
6.4	Mathematical models of turbulence	154
6.4.1	Turbulent viscosity or eddy viscosity models	156
6.4.2	The two equation ($K\sim\epsilon$) model of turbulence	157
6.5	Steady flow simulation for the variable ramp throttling system	158
6.5.1	Grid arrangement and geometric configuration	159
6.5.2	Incorporation of boundary conditions	160
6.5.3	Convergence and grid independence	162
6.6	Steady flow simulation in the butterfly valve throttling system	163
6.6.1	Grid arrangement and geometric configuration	163
6.6.2	Incorporation of boundary conditions	165
6.6.3	Convergence and grid independence	165
6.7	In-cylinder steady flow simulation	166
6.7.1	Grid arrangement and geometric configuration	166
6.7.2	Incorporation of boundary conditions	167
6.8	Discussion of results	168
6.8.1	Ramp throttle cases	168

6.8.2	Butterfly valve cases	172
6.8.3	In-cylinder cases	174
6.9	Concluding remarks	175
	Flow simulation figures	177
CHAPTER 7: COMPUTER MODELS OF IN-CYLINDER UNSTEADY FLOW		
7.1	Introduction	203
7.2	Source of experimental data	204
7.2.1	Experimental model engine configuration	204
7.3	Two-dimensional unsteady simulation	205
7.3.1	Use of moving grids for simulation of engine motion	205
7.3.2	Geometric specifications and grid arrangements	206
7.3.3	Incorporation of boundary conditions	207
7.3.4	Convergence and grid dependence	208
7.4	Discussion of results	208
7.5	Concluding remarks	212
	Unsteady flow simulation figures	214
CONCLUDING REMARKS		232
REFERENCES		235
APPENDIX A		A1

NOMENCLATURE

a_T	Hot wire overheat ratio
A	Crank radius
A_{pis}	Piston area
B	Crank shaft angular speed
C	Connecting-rod length
C_μ, C_1, C_2	Constants in approximate turbulent transport equations
C_{mass}	Coefficient of mass in PHOENICS
C_ϕ	Coefficient of variable in the source terms in PHOENICS eqns.
C_s	Swirl coefficient
d	Hot wire diameter
D_{in}	Inlet diameter of the throttling device
e	Voltage
E	Constant in near-wall description of velocity profile
f_c	Pressure-loss coefficient
f_A, f_V	Porosity factors
h	convective heat transfer coefficient
I	Turbulence intensity
K	Kinetic energy
K_1, K_2	Linearization constants
K_f	Thermal conductivity
l	Macro-scale of turbulence
$\dot{m}$	Mass flow rate
N	Engine speed
NX, NY, NZ	Number of cells in x, y and z-direction respectively
Nu	Nusselt number
P	Pressure

Pr	Prandtl number
q_c	Heat loss by convection
$\dot{Q}$	Average engine flow rate
r	Volume fraction
R	Electrical resistance
Re	Reynolds number
S	Heat transferred surface area
S_ϕ	Source of the dependent variable
t	Time
T	Temperature
$\vec{u}$	Velocity vector
$\tilde{u}$	Relative velocity
U_p	Instantaneous piston velocity
U, V, W	Velocities in x, y and z-direction
V_o	Anemometer output voltage at zero flow
V_{CTA}	Anemometer output voltage
$\bar{V}_{pis}$	Mean piston velocity
V^+	Tangential velocity in logarithmic law of the wall
V_{mass}	Value of mass with physical significance of an external pressure
V_ϕ	Value of variable in the source terms
x, y, z	Orthogonal coordinate directions
y^+	Normal distance in the logarithmic law of the wall
Z_p	Instantaneous piston displacement
α	Temperature coefficient of resistivity
β	Butterfly valve angle
θ	Variable ramp included angle
k	Von Karman's constant
η	Non-dimensional coordinate

ε	Rate of dissipation of turbulence kinetic energy
μ	Molecular viscosity
μ_t	Turbulent viscosity
μ_{eff}	Effective viscosity
ν	Kinematic viscosity
ϕ	A generalized dependent variable
ρ	Density
Γ	Exchange coefficient
$\delta_\varepsilon, \delta_K$	Constants in approximate turbulent transport equations
D/D_t	Mass convection operator

CHAPTER 1I N T R O D U C T I O N

^ A key element in the continued development of gasoline engines which must offer reliable ignition and rapid flame propagation is that of mixture preparation. Homogeneity of fuel-air mixture and accompanying turbulence levels become increasingly important as the attractions of near-limit combustion are explored on the grounds of better fuel economy and reduced pollutant emissions. Improved pre-mixing of charge by turbulent processes in the induction manifold of the engine is one approach to the challenges posed.

Computational models of engine performance have over many years established an increasingly important role in the design process. The focus for these studies, until comparatively recently, has been largely that of synthesis -the assembling of component models to simulate complete engine operation. The formulation of individual models, whether of the induction system or combustion chamber, has been constrained both by the application - which limited range of variables are most appropriate to the overall simulation - and by the complexity of the processes involved, which are typically spatially inhomogeneous and non-stationary. A new generation of computer-based methods is now emerging for calculating the detailed patterns of gas flow, heat transfer and combustion in reciprocating engines by solving numerically the governing partial-differential conservation equations for fluid motion. The recent developments in computational fluid dynamics have made much of the fine detail of the engine aerothermochemistry accessible and so offer the component designer a more finely-resolved diagnostic tool. It is on the computer model in this last role, in relation to induction systems and their effect on in-cylinder flow, that the present thesis concentrates.

Whilst wave propagation models of induction systems, employing the method of characteristics for example (cf. Horlock and Winterbone (1986)), address many important questions in relation to transient

gas dynamics, they are quite uninformative on the turbulent mixing process, the effective use of which is an increasingly prominent feature in engine gas path design. Variations in the method of flow restriction - alternatives to the butterfly valve - provide one method of approach which has been investigated (Ma (1975), Beale and Hodgetts (1976)) and found to offer performance improvements in respect of weak mixture combustion and hence part-load fuel consumption and emissions levels. However, internal flows generally offer only limited access for detailed diagnostic measurement and inferences regarding mechanisms are often difficult to substantiate from exhaust and restricted in-cylinder observation. As both confidence in the reliability of computational flow-field simulation and its availability have grown it becomes an increasingly powerful element in the evaluation of alternative designs.

This thesis describes the detailed computer simulation of the steady flow through two contrasting variable geometry restrictions using the PHOENICS computer code. These comprise a triangular ramp of variable included angle in a substantially rectangular duct and a conventional butterfly valve. Overall flow field features are compared with the results of water analogy flow visualization and detailed predictions with limited point measurements using a single component hot-wire anemometer. This work also sets wider objectives in that it describes a program of research aimed at quantitatively assessing the capabilities of a method of this kind for the prediction of unsteady gas flows in motored engines.

In Chapter 2 key features of the turbulent flow in spark ignition engines are reviewed during the intake and compression processes. Additionally this chapter seeks to identify the influence this turbulent field has on the combustion process.

In Chapter 3 a detailed review is described of the attempts made over the past few years to improve lean mixture combustion as a method of reducing exhaust emissions and obtaining better fuel economy. The effects of squish, swirl and high compression ratios on combustion efficiency and in-cylinder charge motion is briefly assessed. More immediately relevant to this research, a number of alternative throttling devices are presented and their influence on induction manifold mixture distribution and mixing is appraised.

Broader aspects of the investigative methods and approaches relevant to fluid motion within internal combustion engines are briefly discussed

In Chapter 4 the water analogy studies of induction flows through clear Perspex models at full-scale are presented. Water containing finely dispersed air bubbles, pumped through the models and illuminated by a light sheet, provides a clear picture of these complex flow fields. The resulting flow visualization images are used as a means of direct comparison for the gross features of the flows and, more importantly, for validation and assessment of the computer simulated flows in later sections.

In Chapter 5 the largely qualitative data obtained from the water analogy tests of the preceding chapter are complemented by hot-wire velocity measurements at particular stations, with air replacing water as the flow medium in the same Perspex models.

In Chapter 6 an extensive computational simulation of steady flows through the induction tracts of the alternative systems is reported. The flexibility of the code (PHOENICS) for simulation of three-dimensional elliptic flows is exploited in representations of the model geometries. The computer predicted flow fields are assessed and compared with the available experimental data. A critical criterion in terms of induction flow distribution, mixing and turbulence is identified, which offers, at last in part, a plausible explanation for some of the effects of alternative induction systems observed in firing engines. In addition, an outline is provided of the essential features of the computational method and the principal sources of uncertainty which can influence its development.

In Chapter 7 a program of research aimed at quantitative assessment of the capabilities of computational methods for the prediction of gas flow in motored engines (i.e. transient flow) is presented. Comparisons are shown between PHOENICS predictions and experimental measurements of turbulent gas flow in a non-compressing axisymmetric model engine with annular inlet port reported in the literature. Suggestions are made to improve the modelling of the entering annular jet.

A summary of the work reported in this thesis, together with proposals for future related research, is presented in Chapter 8.

CHAPTER 2

TURBULENCE AND TURBULENT COMBUSTION IN SPARK-IGNITION ENGINES

2.1 Introduction

The turbulent flow field in an engine plays an important role in determining its combustion characteristics and thermal efficiency. Automotive engineers have learned that changes in combustion chamber shape and inlet system geometry, both of which change the turbulent flow field, influence emissions, fuel economy and lean operation limits of an engine.

Today the cost of an engine development program can no longer be measured only by the capital outlay in equipment but also by the number of months and years before an engine concept can be demonstrated either successfully or unsuccessfully. In an effort to shorten engine development times researchers have developed analytical techniques to simulate engine operation and to predict the detailed performance of spark-ignition engines. Among other factors details of turbulence structure in the engine are needed for determining such characteristics as heat transfer rates, ignition delay times, minimum ignition energy and the rate of mixing and burn-up of quench layers.

In this chapter an attempt is made to review key features of the turbulent flow in an engine during intake and compression processes and to identify the influence this turbulent field has on the combustion process.

2.2 Turbulent flow field in a spark-ignition engine

2.2.1 General features of the turbulent flow field

The general features of the turbulent flow field in an engine can be represented by mean flow velocity and turbulence intensity as shown in Fig. 2.1. One important feature of the flow is the large magnitude of mean velocities and turbulence intensities generated

during the intake process. The high velocities in the intake process generate a highly turbulent flow and are the main source of turbulence that persists throughout the compression and expansion phases of the engine cycle. Another general feature of the flow is the rapid decay of both mean flow and turbulence intensity at the end of the induction stroke as the piston velocity approaches zero and the intake valve closes. This occurs since there is little or no further turbulence production at this phase in the cycle and the viscous dissipation time for the turbulent kinetic energy is of the order of a millisecond.

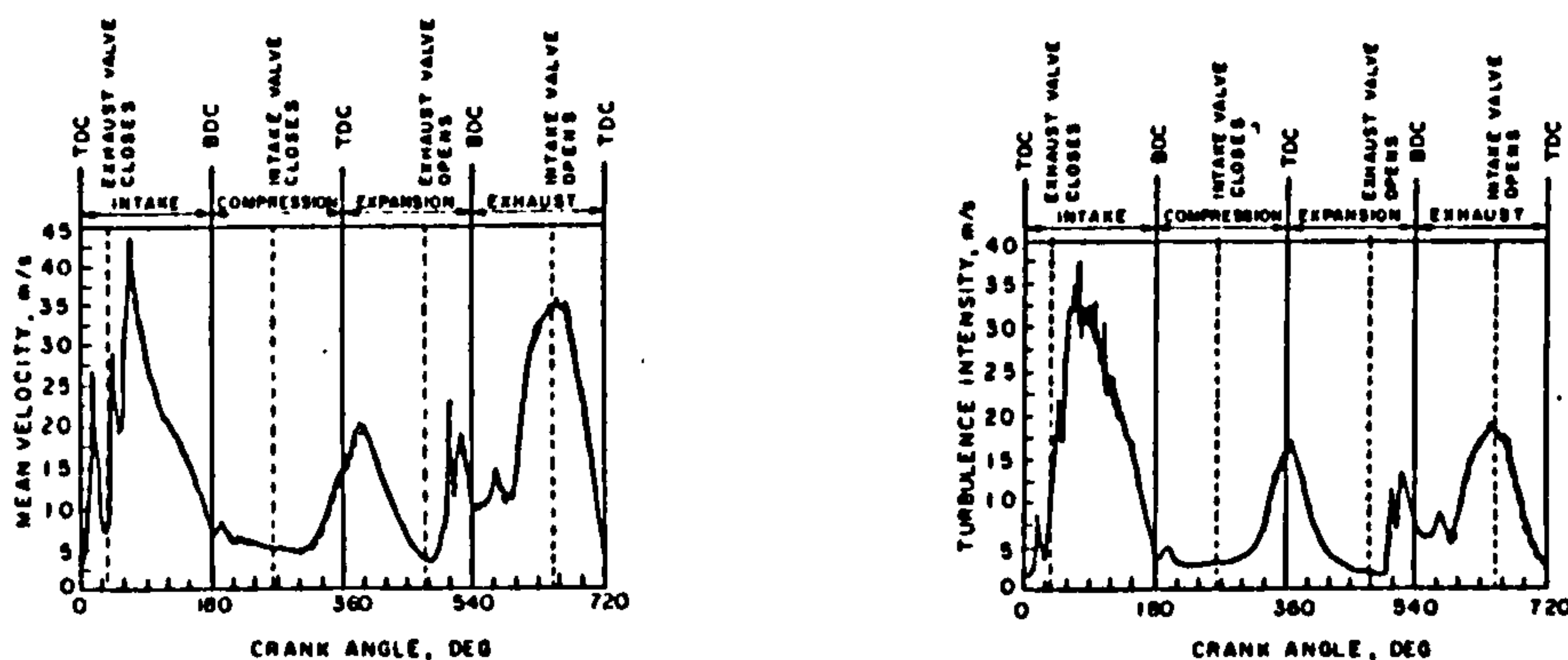


Fig. 2.1 Turbulence intensity and mean velocity averages over 1000 cycles. (From Ref. 1)

The compression and expansion are highly dependent on combustion chamber shape. For disk chambers with no squish or swirl the dissipation process generally continues but at a much slower rate than during intake. For chambers with squish or swirl the structure of turbulent field is similar to that shown in Fig. 2.1. The squish region generates high velocities when the piston approaches TDC and there is turbulence production resulting in an increase in the turbulence intensity.

During the expansion phase turbulence decay processes again predominate and continue up to the time of exhaust valve opening. As the exhaust valve opens a boundary layer sink flow develops and high mean flows and turbulence levels are observed. These appear

as peaks in velocity which occur during exhaust, an initial peak as the exhaust valve opens, and a second peak near maximum piston velocity.

2.2.2 Intake process

The jet flow through the inlet valve of an engine during the intake process is generally thought to be mainly responsible for the production of the turbulent field in the engine cylinder. In an experiment by Semenov (2) in a motored engine a rapid decay in instantaneous velocity was shown as the intake and exhaust process were eliminated (cf. Fig 2.2). Semenov has also made extensive measurements at different points in the chamber during intake. His

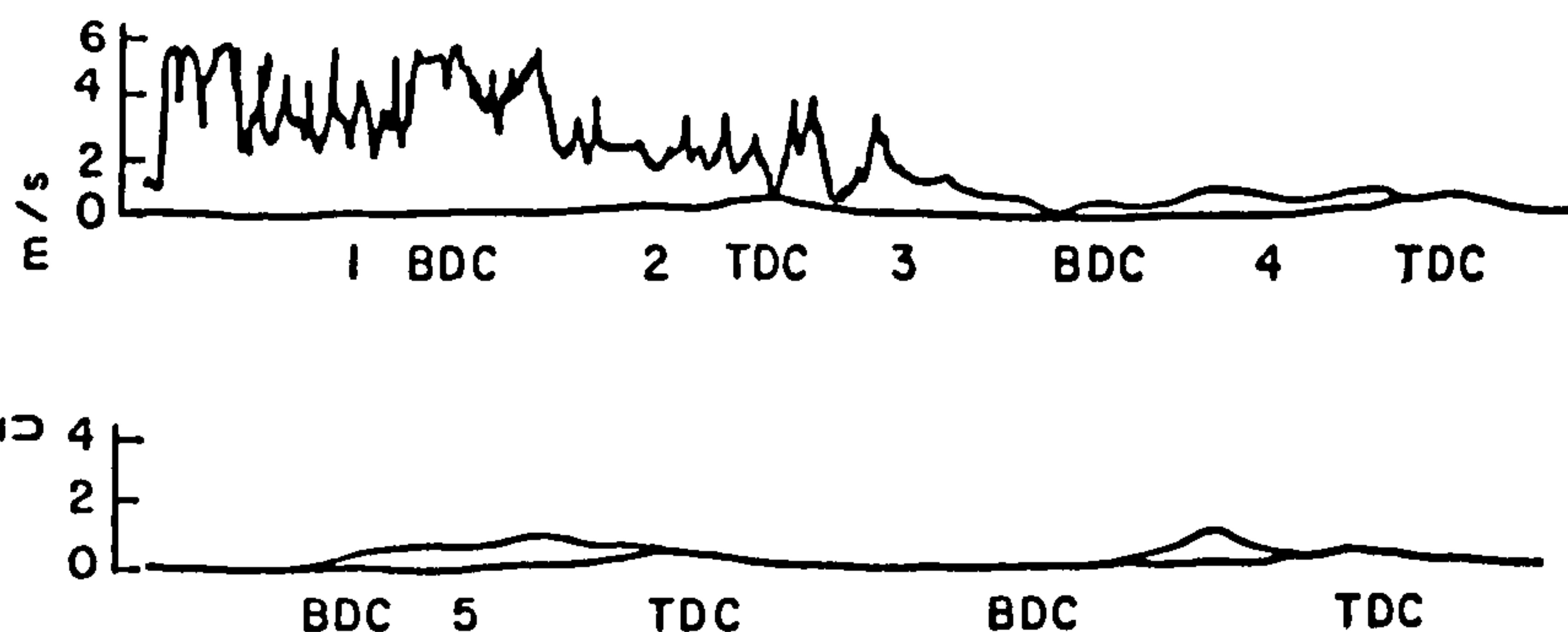


Fig. 2.2 Oscillogram trace of velocity during transition from normal cycles to a cycle in which intake and exhaust are absent. (From Ref. 2). (1) Intake; (2) compression; (3) expansion; (4) second compression; (5) third compression.

measurements of mean velocity during intake at various locations in the cylinder are shown in Fig. 2.3. These results show the presence of large velocity gradients across the chambers and suggest the jet-like nature of the intake flow process. The existence of these

large mean velocity gradients also implies that the intake process is highly anisotropic.

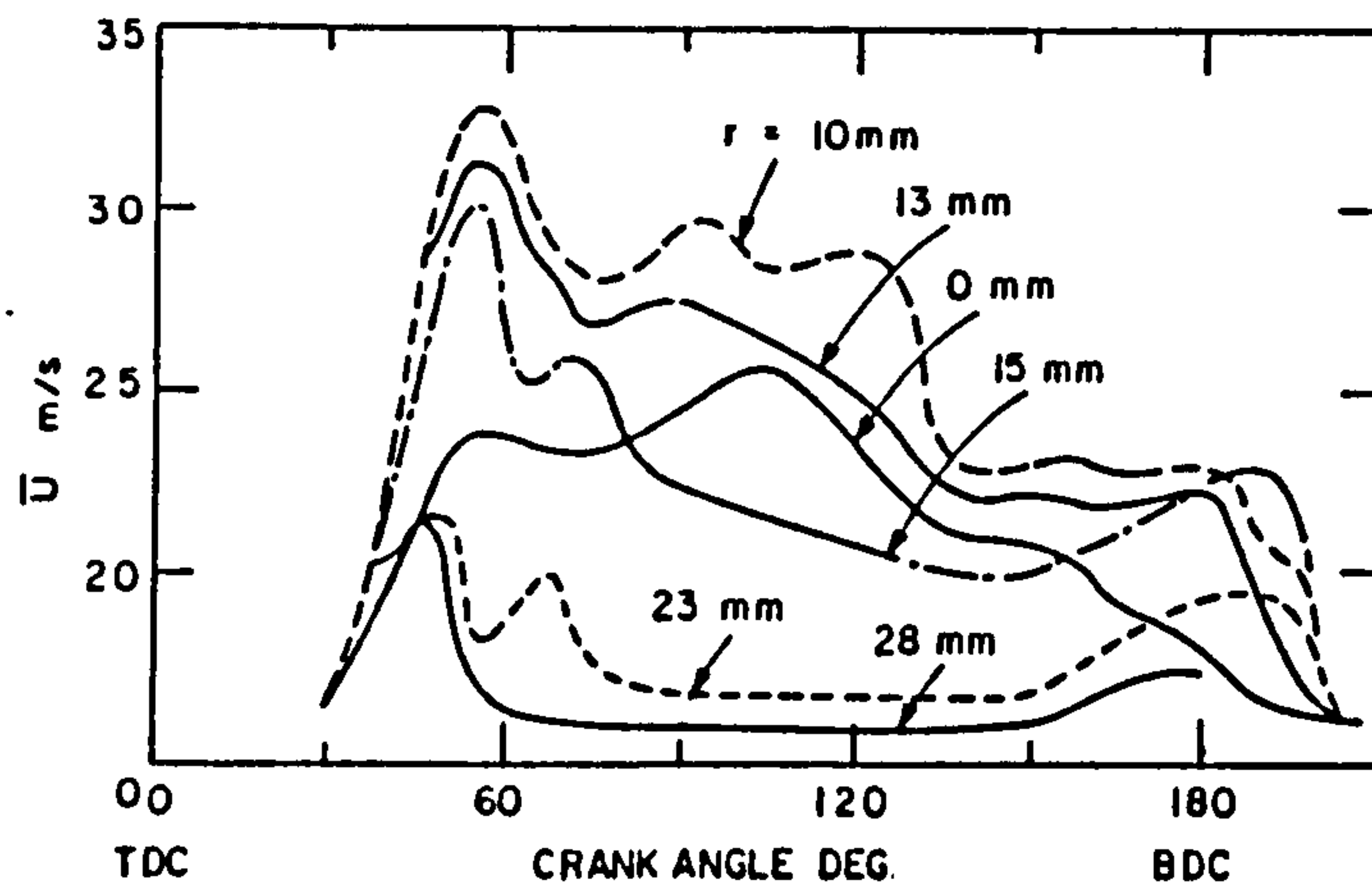


Fig. 2.3 Variation of mean velocity during intake at various points in the chamber. (From Ref. 2)

2.2.3 Compression process

Combustion occurs in the vicinity of TDC during the compression process. As mentioned previously, the flow field in the vicinity of TDC is highly dependent on combustion chamber shape and inlet swirl.

During the early portions of the compression process the turbulent flow field is still dominated by the intake process. The mean velocity is decaying rapidly since little energy is being supplied to the flow. Similarly the turbulence intensity decreases. These features are depicted in Fig. 2.4 (3), which show some typical results for the compression process. These results show that during the early portions of compression, combustion chamber shape, inlet swirl and valve effective area are only important in establishing the initial levels of turbulence which the decay processes then diminish.

As TDC is approached the flow field is influenced by the inlet swirl and the squish region in the combustion chamber. If no swirl or squish is present then the mean velocity tends to level off and

then rise slightly near TDC (cf. Fig. 2.5 (2)), since there is no turbulence production except due to the piston motion. In geometries

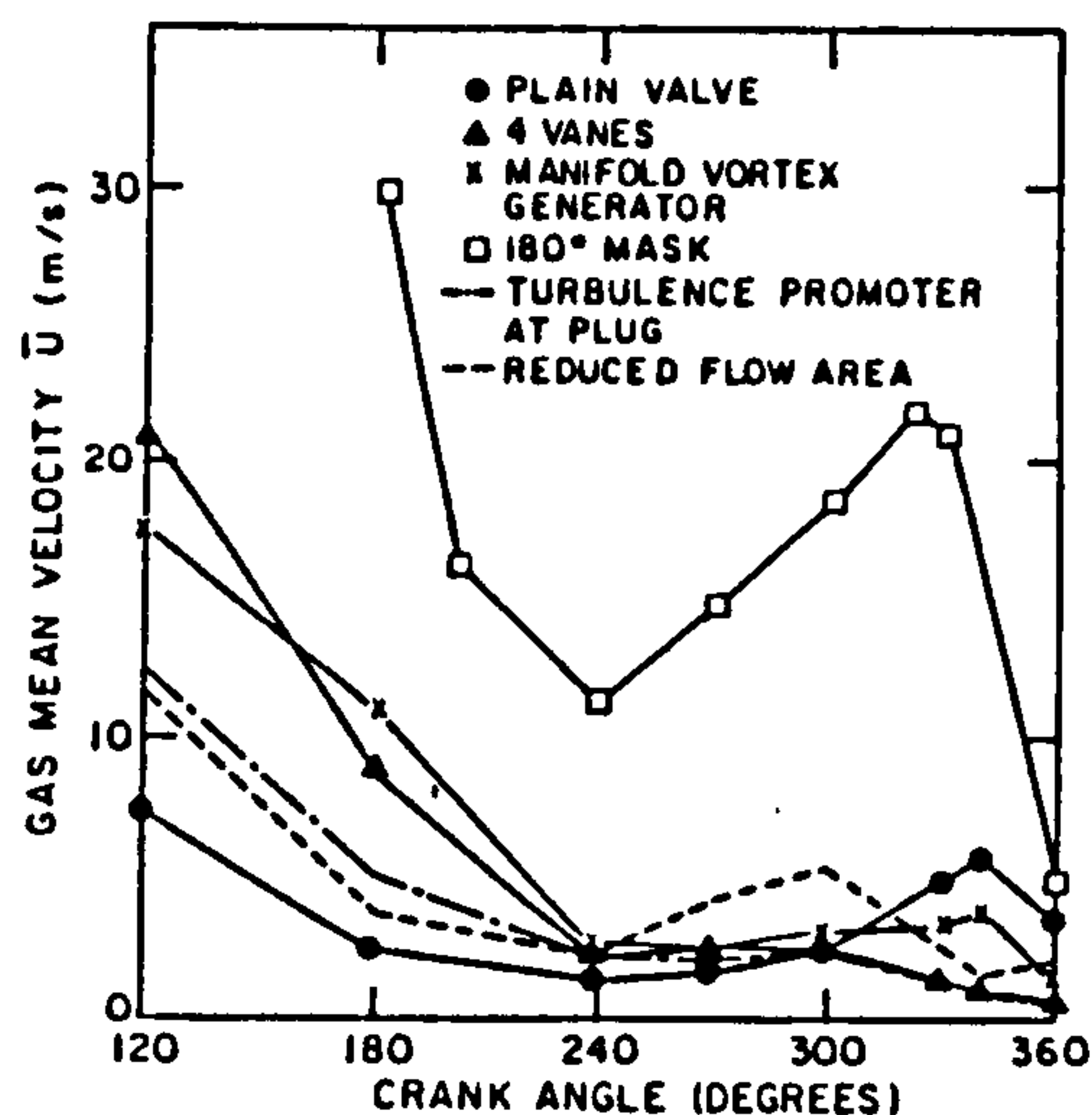


Fig. 2.4 Effect of various turbulence promoters on velocity variation for a wedge chamber at WOT 1000 rpm (From Ref. 3).

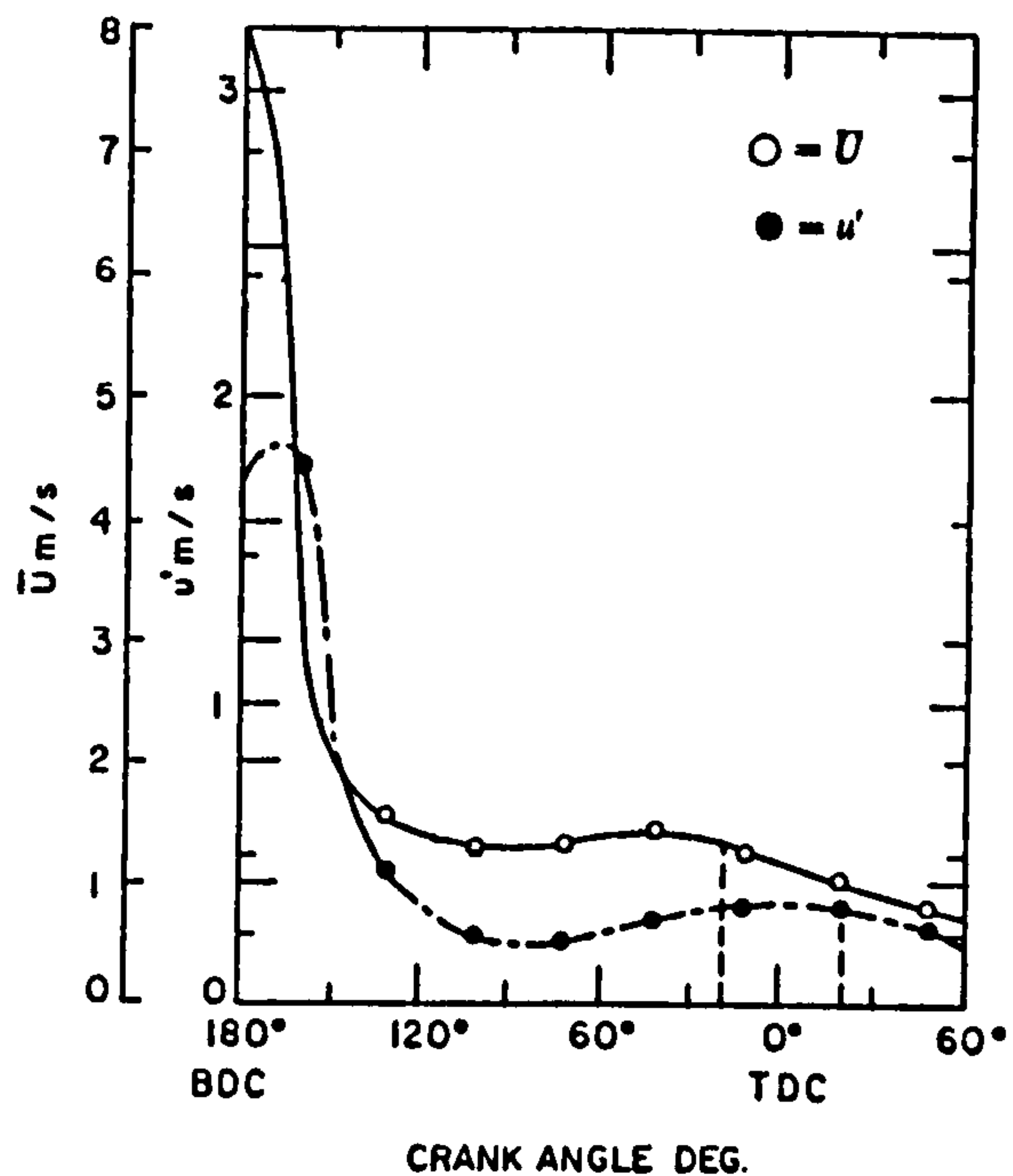


Fig. 2.5 Average and fluctuating velocities during intake.
(From Ref. 2)

where either large squish regions or high swirl rates exist, the turbulence produced by these techniques is much higher than that due to mean flow produced by piston motion.

The effect of swirl and squish on mean velocity and turbulence intensity can be seen in Figs. 2.4 and 2.6. Near the TDC the effects of both parameters are to increase the mean velocities and turbulence levels. Although the production of mean velocities and turbulence intensities near TDC due to squish is obvious, this result for swirling flow is not straightforward. If the swirling motion was similar to solid body rotation no variations in the mean velocity should occur. It is evident from the results of Dent and Salama (3), Lancaster (4) and Meyer and Birnie (5) that significant solid body rotation

does not exist and swirl motion must be highly non-uniform. The compression process should reduce this non-uniformity.

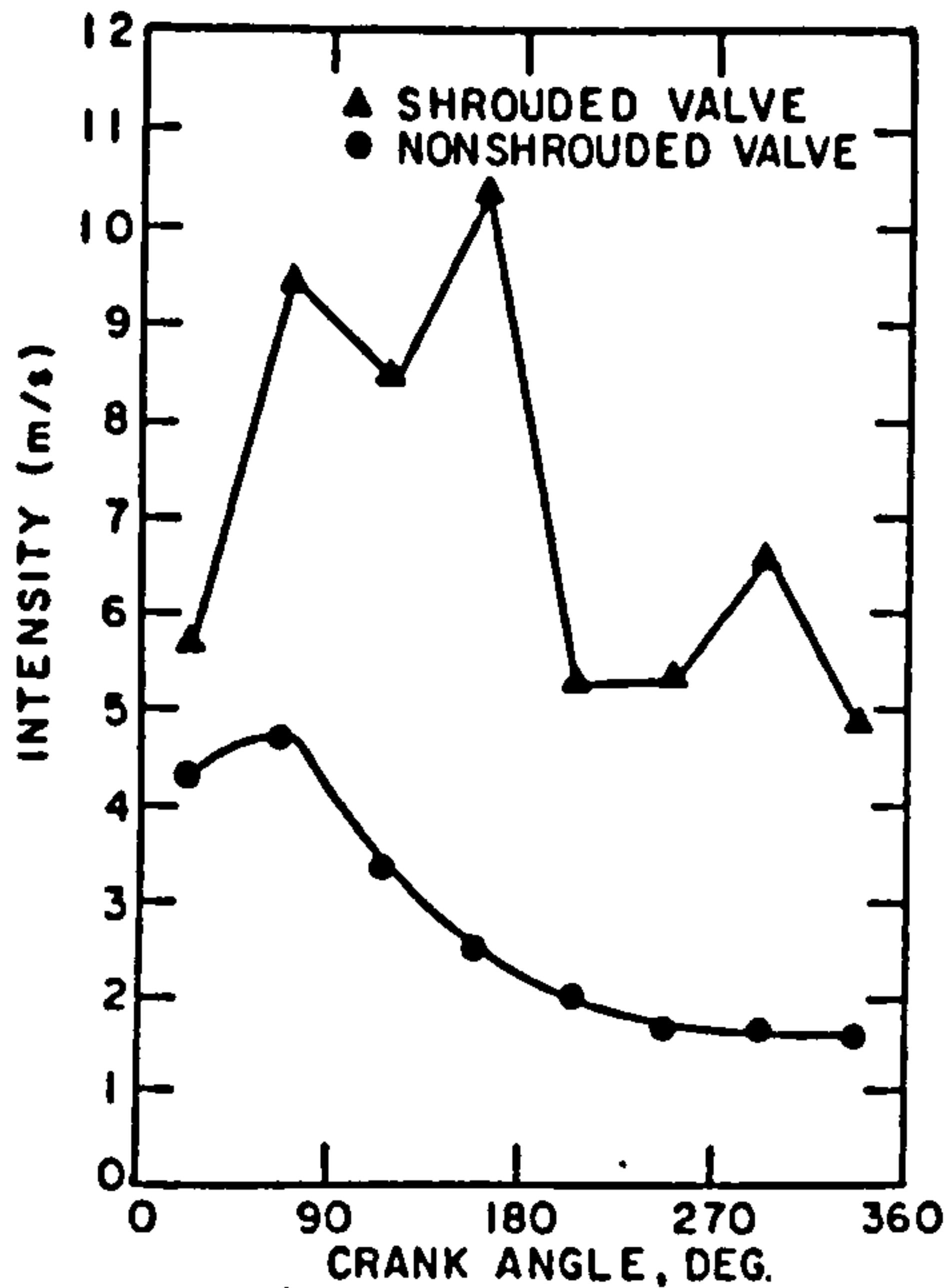


Fig. 2.6 Turbulence intensity during intake and compression for shrouded and non-shrouded valve geometries. (From Ref. 4)

2.3 Effects of engine variables on turbulence in spark-ignition engines.

2.3.1 Engine speed

Since an engine operates over a wide range of conditions - engine speed, load and compression ratio- it is important to understand how turbulence quantities vary with these operating variables. Most investigations have been concentrated on engine operating variables at or near TDC of the compression stroke.

The most universal result is the linear variation of turbulence intensity and mean velocity with engine speed, which is independent of combustion chamber shape and inlet swirl, as can be seen in Figs. 2.7 and 2.8.

The variation of turbulence length scale with engine speed is shown in Fig. 2.9. For the non-shrouded valve the integral length

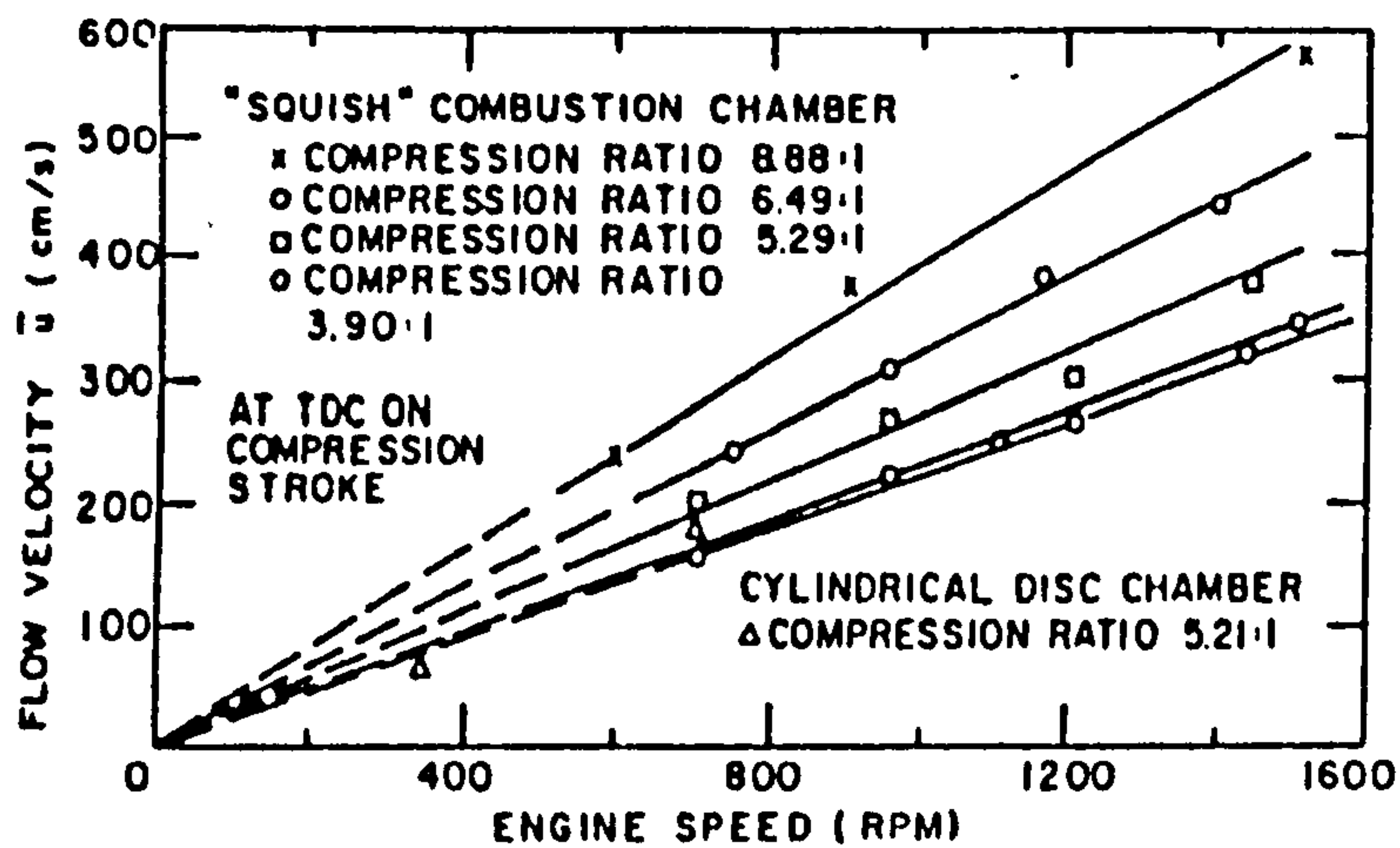


Fig. 2.7 Mean flow velocity versus engine speed. (From Ref. 6)

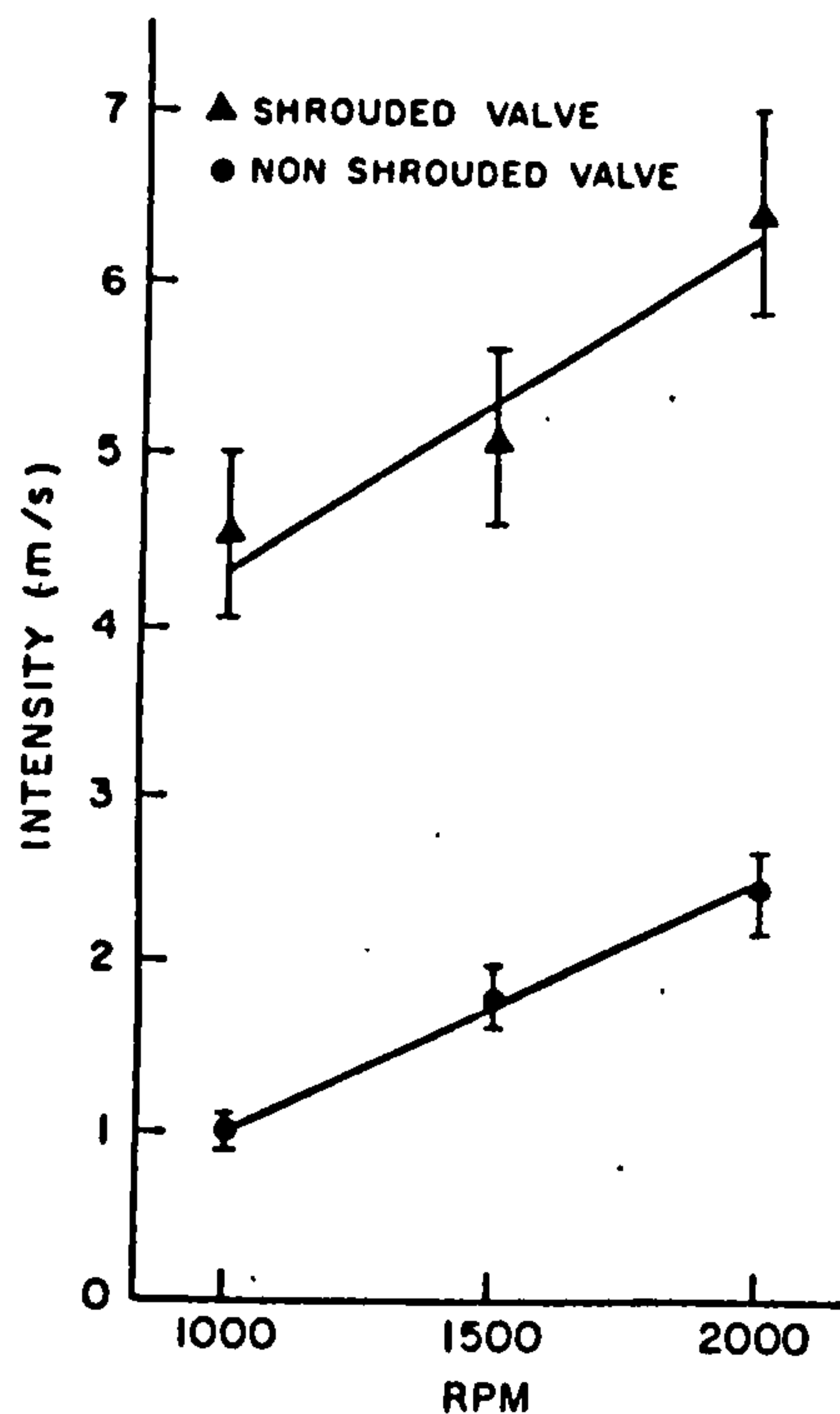


Fig. 2.8 Turbulence intensity versus engine speed. (From Ref. 4)

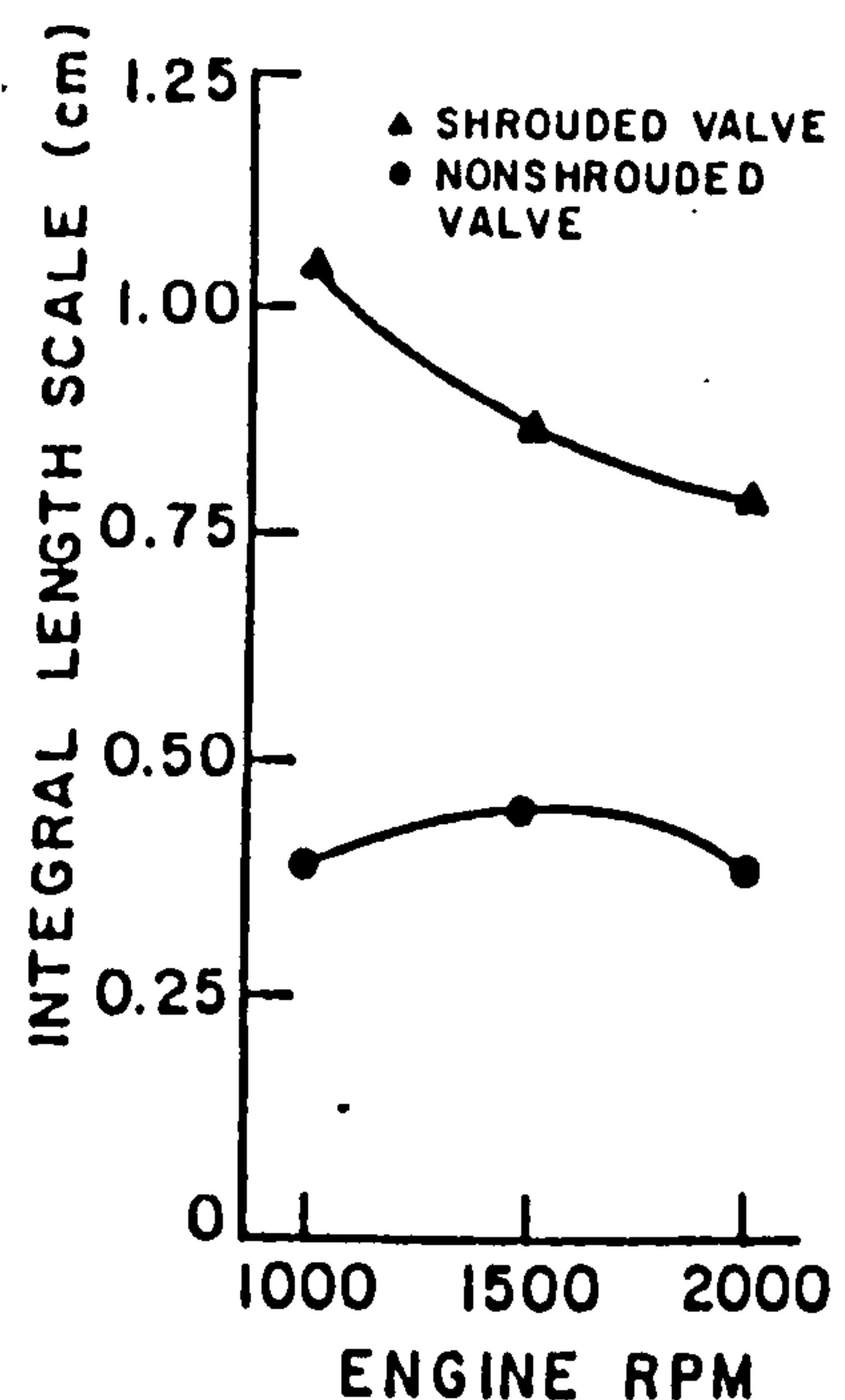


Fig. 2.9 Variation of turbulence integral length scales with engine speed. (From Ref. 4)

scale is not dependent on engine speed. For the shrouded valve the integral length scale decreases with engine speed, although the dependence is small. If the turbulent flow field is relaxed then the integral scale should be of the order of the height of the chamber. These results indicate that the flow has relaxed since the dimensions of the chamber are controlling the turbulence scale.

2.3.2 Engine compression ratio

From the previous section, one would expect the integral scale of turbulence to decrease as compression ratio is increased if the flow is relaxed. Dent and Salama (7) show such a decrease in Taylor microscale as compression ratio is increased. In a study of ignition delay Blizzard and Keck (8) presented a correlation for change in turbulence scale as a function of distance from cylinder head. This correlation can be written in terms of compression ratio and is given as

$$\lambda = 0.17 (\text{valve lift}) / (\text{compression ratio}) \quad (2.1)$$

Although Keck and Blizzard did not explicitly identify this scale as the Taylor microscale, Dent and Salama's (7) results for Taylor microscale as a function of compression ratio show good agreement with equation 2.1

2.3.3 Engine volumetric efficiency

When the intake pressure and density are reduced the flow velocities through the intake valve decrease and the volumetric efficiency of the engine decreases. This essentially reduces the amount of kinetic energy supplied to the flow. One would expect that the turbulence intensity would increase as load increases, as shown in Fig. 2.10. This effect is not large for the non-shrouded valve, low turbulence chamber since there is ample time for dissipation of kinetic energy before TDC of the compression stroke. However, the organized motion of swirl decays much more slowly and hence shows a much larger dependence on volumetric efficiency.

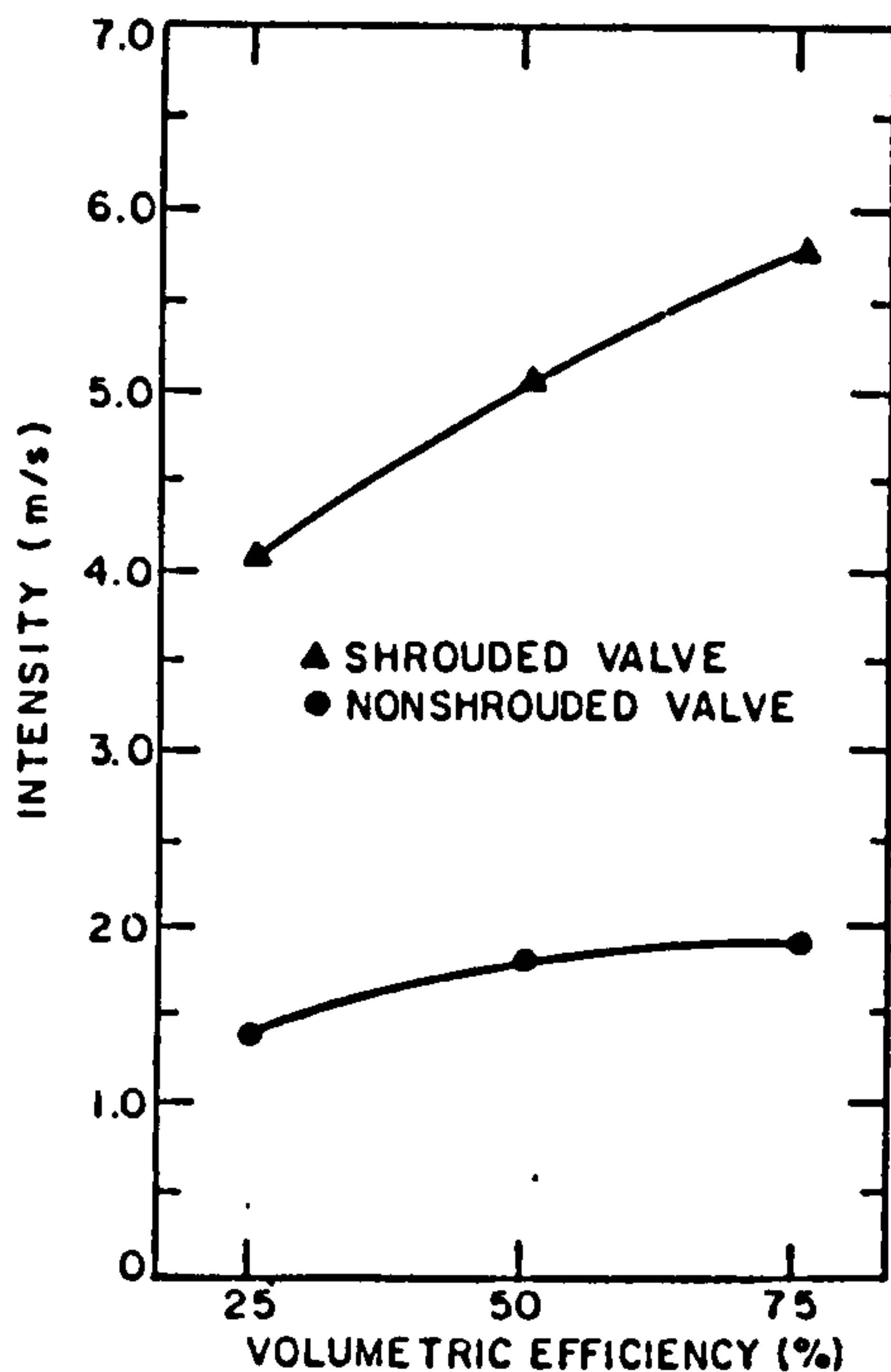


Fig. 2.10 Variation of turbulent intensity with volumetric efficiency.
(From Ref. 4)

2.4 Turbulent combustion and flame propagation in the spark-ignition engine.

2.4.1 Turbulent flame propagation

A flame is a rapid, self-sustaining chemical reaction occurring in a discrete reaction zone (15). In non turbulent mixture, flame propagation is laminar and the flame has a smooth surface and relatively thin reaction zone. The flame speed, or speed with which reaction zone moves relative to unburned mixture, is determined by the chemical and thermodynamic properties of the mixture. When turbulence is present the flame front is no longer smooth and the reaction zone is thicker than in the laminar case. In addition, the flame speed when turbulence is present is several times the laminar value, depending on the intensity of turbulence.

There are two mechanisms which have been used to explain the increase in flame speed due to turbulence. The first mechanism considers

the effects of turbulent eddies of a scale less than the thickness of the laminar flame front. These eddies are assumed to increase the local heat and transfer rates along the flame front, thereby increasing the local rate of flame propagation. The second mechanism used to explain the increase in flame speed due to turbulence is generally accepted as the more important, and considers the effects of turbulent eddies of a scale larger than the thickness of the flame front. These eddies are assumed to have no effect on the local flame velocity, but to distort the flame front so that its area is increased. The increase in flame speed is then proportional to the increased area of the flame front.

The assumption that turbulent scale affects the mechanism by which turbulence influences flame propagation suggests that turbulent scale should influence flame speed. Many experimental studies have found, however, that turbulence intensity is the only characteristic of turbulence which influences flame speed (15, 16). This is probably because the energy of turbulent flow is not contained in a single eddy size, but is distributed over a continuous range of eddy sizes.

2.4.2 Turbulent combustion process in spark-ignition engines

The turbulent combustion process in a spark-ignition engine can be characterized by a plot of flame front position as a function of time. Fig. 2.11 shows such a plot depicting the various regimes of the combustion process. The initial phase of the combustion process

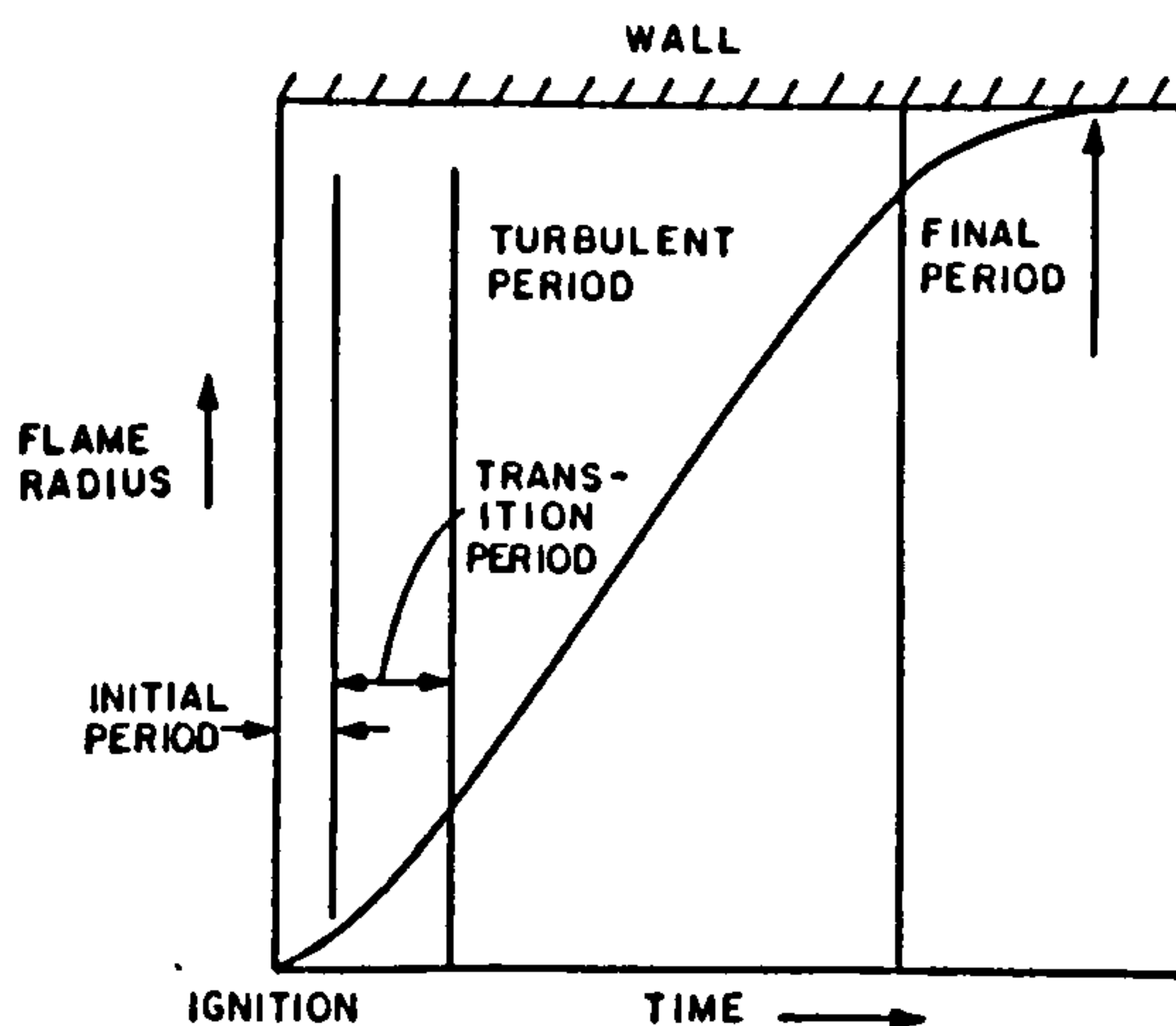


Fig. 2.11 Average flame front position as a function of time.

is characterized by its dependence on local turbulent field at the spark plug at the time of ignition. This initial phase continues until several eddies, the size of the chamber height, are fully burned. Variations in this initial burning process are responsible for much of the cyclic variation observed in spark-ignition engines. The fully developed turbulent phase of the combustion process takes place over most of the chamber and is the phase in which most of the charge is burned. This phase of the combustion process can be analyzed by the average turbulence quantities since the flame front is entraining many eddies at the time as it progresses across the chamber. For this phase, the turbulent fluid mechanics, or in other words interaction between turbulence field and heat release, is the governing mechanism for flame propagation. A final combustion phase after most of the charge is burned is also present and is dependent both on the local turbulence structure just ahead of the flame front, wall effects and also on the rate of chemical reaction especially at lean conditions.

2.4.3 The influence of turbulence on ignition

Before the turbulent flame propagation process can begin the existence of a viable flame kernel must be established. For this reason it is of interest to know the effect of turbulent intensity and scale on the ignition process. Although the process of ignition has been studied widely, the effect of turbulence on ignition has been examined by relatively few investigators (17-20). In general, theoretical analyses of ignition process are a result of balancing the heat generated by ignition and combustion against the heat loss due to heat conduction to the unburned gases. The experimental work that has been performed has been mainly restricted to low pressure, flowing gas streams or combustion bombs, with approximately isotropic turbulence. Although these flow processes that are studied do not simulate those in an engine, some insight into trend behaviour of the ignition process is possible. The experimental results of Ballal and Lefebvre (9) illustrate the influence of turbulence on the minimum ignition energy (Fig. 2.12). These results show the minimum ignition energy is a strong function of turbulence intensity and equivalence

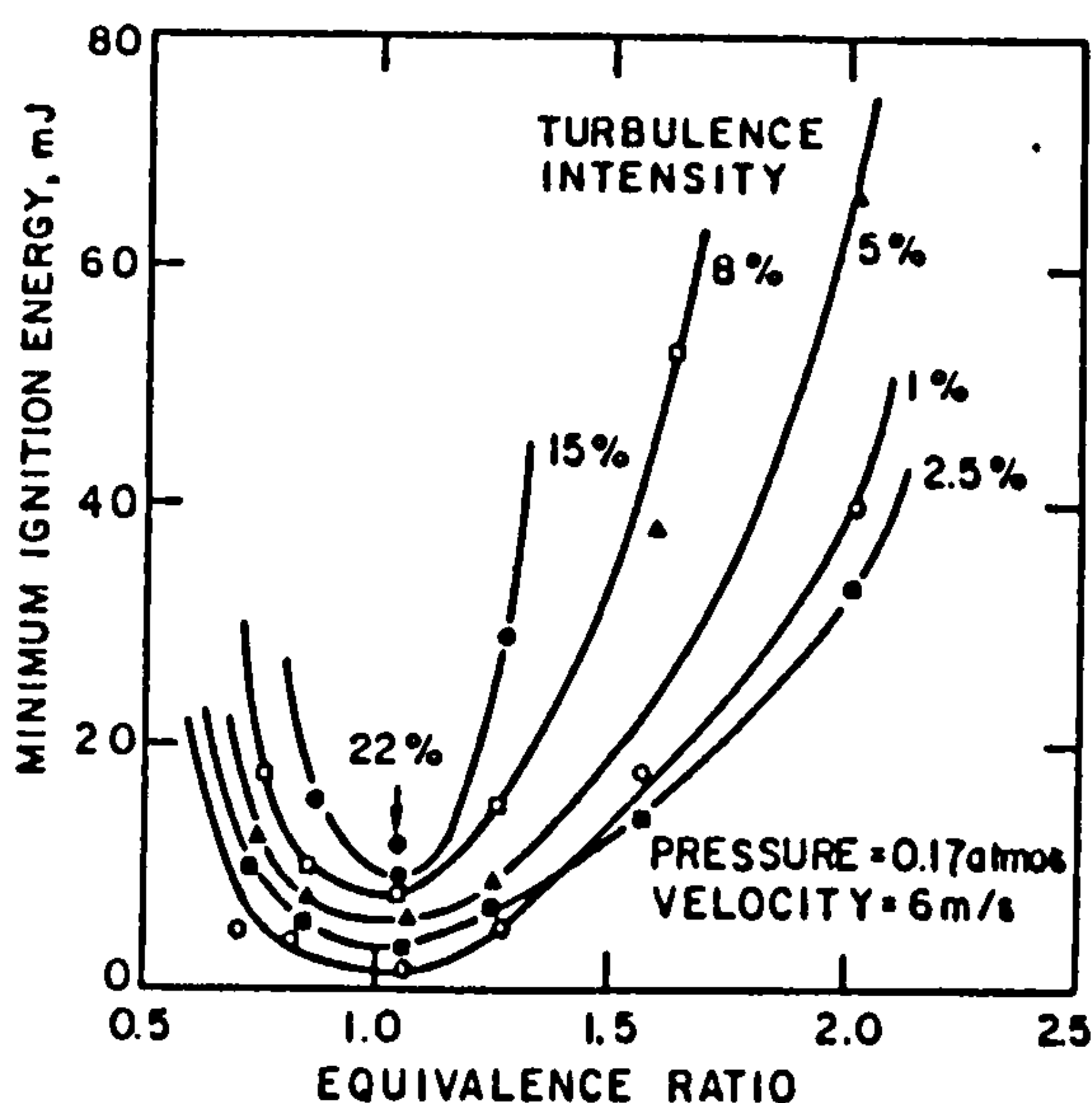


Fig. 2.12 Effect of relative turbulence intensity on minimum ignition energy .
(From Ref. 9)

ratio. The influence of turbulence is increased as the stoichiometry deviates from the chemically correct value. Several general conclusions on the the influence of turbulence on ignition can be stated.

a) The presence of turbulence increases the average reaction zone thickness which in turn increases the critical volume into which the ignition energy must be released.

b) The presence of turbulence increases the rate of heat dissipation which results in a decrease of energy available for ignition.

c) Spark duration and distribution of energy input strongly influence the critical ignition energy.

2.4.4 Influence of turbulence on ignition delay

Ignition delay in a spark-ignition engine is generally defined as the time from spark initiation to the time when 10% of the mass is burned. The phenomenon of ignition delay has been the subject of many investigations and it has been shown that variations in the ignition delay time are the major cause of cycle to cycle variations in the cylinder pressure (10). The variations in ignition delay time can be caused by variations in velocity near the spark plug and mixture non-homogeneities (11). Overall it is well accepted that cycle to

cycle variations in the turbulent field near the spark plug result in cycle to cycle variations in the cylinder pressure. Several investigators (11, 12) have sought to quantify this effect by measuring the velocity field at the spark plug and correlating standard deviations of the velocity with peak pressure and the angle of occurrence of peak pressure. The correlation coefficients were of the order of 0.97, showing a high degree of correlation between cycle to cycle variations of velocity and pressure.

2.4.5 Influence of turbulence on flame propagation in engines

The fully developed portion of the turbulent flame propagation process in an engine -simple geometries, relatively rich mixtures- (i.e. 10-90% burned) has been observed to scale almost linearly with engine speed. This fact, combined with the observation that the turbulence intensity is also a linear function of the turbulent flame speed, implies that the turbulent flame speed is approximately a linear function of the turbulence intensity. This linear relationship between turbulent flame speed and intensity has been verified in studies of turbulent combustion in bombs (13) and engines (14). Ohigashi (13) and Lancaster (14) also measured the effect of equivalence ratio on the turbulent flame speed. Their results imply that under normal operating conditions the flame speed is only a function of the turbulence intensity. However, as the flammability limits are approached the turbulence scale should play an important role since it is the shearing of the reaction zone that is of primary importance for flame extinction.

2.5 Concluding remarks

There still remain many unanswered questions with regard to the turbulence structure of the flow field in the cylinder of a spark-ignition engine and influence of turbulence on combustion. Nevertheless a number of useful conclusions concerning the turbulence characteristics in an engine cylinder can be stated.

- 1) The high shear flows generated during the intake process

are the major turbulence production source for combustion chambers with small squish and no swirl.

2) The initial turbulent flow field rapidly decays so that the shape of chamber becomes a dominant factor governing the turbulence structure near TDC on the compression stroke.

3) Little or no mean flow exists near TDC for engine combustion chamber geometries with no swirl and squish.

4) There is evidence that the integral scale of turbulence near TDC is of the order of chamber height.

In addition, several statements on the structure of turbulent flame front and its rate of propagation can be made:

1) The turbulent flame has a thickness which is on the order of the height of the chamber.

2) The turbulent burning process takes place on a scale the size of the Taylor microscale. There is evidence that this scale is a dominant variable for the ignition delay process.

3) Under normal operating conditions the fluid mechanics (turbulence intensity) govern the rate of flame propagation.

This chapter primarily dealt with the review of studies, results and correlations which were to a large extent idealized, but nevertheless valuable in providing a better understanding of combustion processes in spark-ignition engines.

Factors such as ignition timing, mixture preparation, combustion chamber design, fuel droplet size and distribution, and cylinder to cylinder and cycle to cycle charge distribution can effect combustion processes severely. Abnormalities such as auto-combustion (better known as spark knock), misfiring and flame quenching are among cases to which idealised approach can not be fully applied.

The following chapter will look at the practical attempts made over the years to overcome some of these problems, and in particular the effect of mixture preparation and distribution on lean mixture running of spark-ignition engines.

CHAPTER 3

REVIEW OF PREVIOUS STUDIES OF LEAN MIXTURE COMBUSTION

3.1 Introduction

A great amount of interest has been shown over the past few years in lean mixture combustion as a method of reducing exhaust emissions and obtaining better fuel economy. This has been mainly prompted by the substantial rise in fuel prices and the strict measures brought in to control exhaust emissions. However, in practice satisfactory lean combustion has been found difficult to achieve. Whilst the initial benefits expected are in fact realised, as mixture is weakened to a considerable extent beyond the stoichiometric air-fuel ratio, specific fuel consumption and hydrocarbon emissions are again increased. This has been found to be largely attributable to slow combustion rates and the failure for a flame to be established consistently, which is commonly referred to as misfiring. Methods have been examined to improve these burning rates so that the weak running can be used extensively and thereby achieve the improved fuel consumption and lower exhaust emissions associated with the lean running. The effects of air-fuel mixture strength on power, fuel consumption and exhaust emission are illustrated in Fig. 3.1.

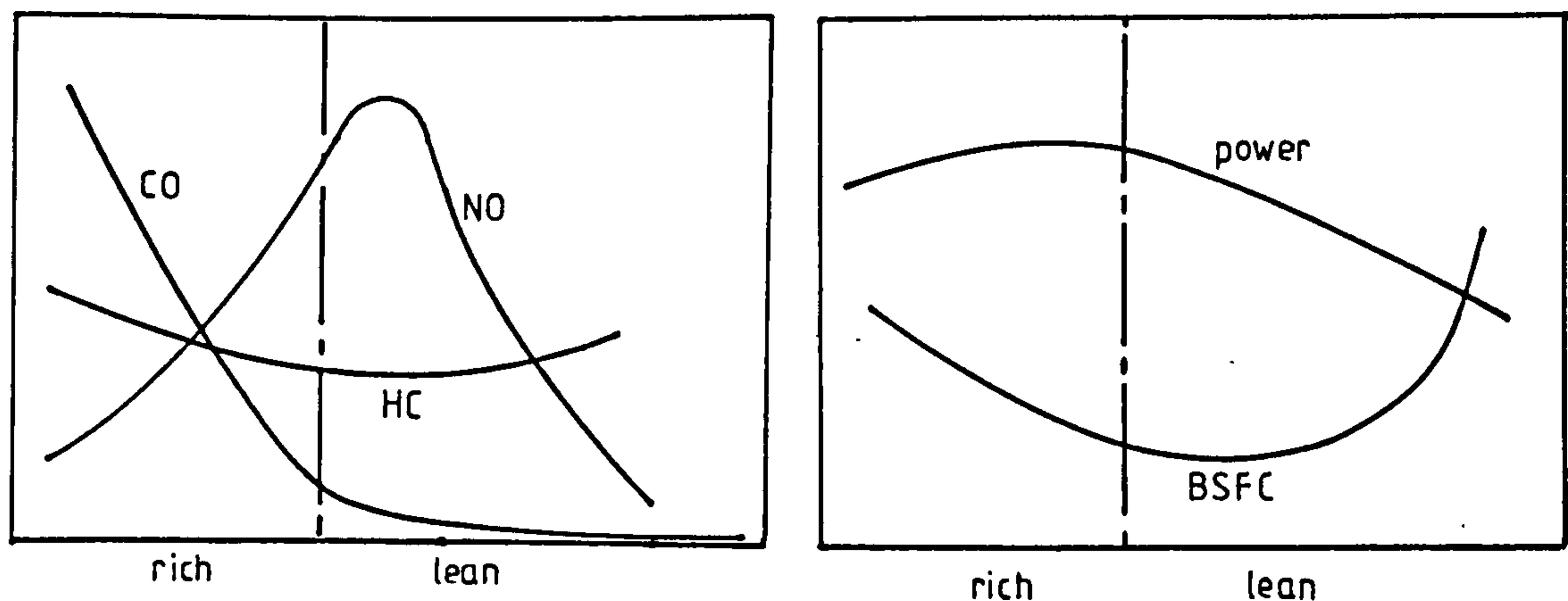


Fig. 3.1 Effect of air-fuel ratios on power, fuel economy and emission.

In addition, the use of higher than normal compression ratios (> 8 , say) has been known to assist the running of engines with weaker than normal fuel-air mixtures. It has been suggested that this is partly due to increased flame velocity as a result of reduced residual mass fraction (20), and the dependency of integral scale of turbulence on combustion chamber geometry, as mentioned in Chapter 2. Theoretically higher efficiencies can also be obtained by increasing the compression ratio. This can be demonstrated by simply calculating the efficiency for various compression ratios based upon the Otto fuel-air cycle with the assumption of constant volume adiabatic combustion and isentropic compression and expansion. However, the practical appreciation of high compression ratios is restricted by excessive spark knock, especially at low speed, high load conditions.

A considerable amount of research has been carried out to determine the effects of mixture preparation on combustion efficiency. This has led to some improvements in the combustion of weak mixtures, as well as extension of lean limit of operation in spark-ignition engines. Particular emphasis in this area has been placed on the production of smaller than normal fuel droplets and mixture motion, including turbulence promotion. This has been widely presumed to ensure a homogeneous mixture delivery to each cylinder and minimise cylinder to cylinder variations.

Based on the conclusions drawn in Chapter 2 it seems plausible to presume that the more nearly homogeneous mixture which can be achieved by producing smaller than normal fuel droplets, the subsequent evaporation of fuel together with intensive turbulent movement of combustion mixture, can improve the combustion efficiency of lean mixtures of fuel and air by increasing the effective combustion rates.

In practice there are two engine regimes in which mixture motion can be generated:

- a) Mixture motion generated in the inlet manifold.
- b) Combustion chamber generated motion.

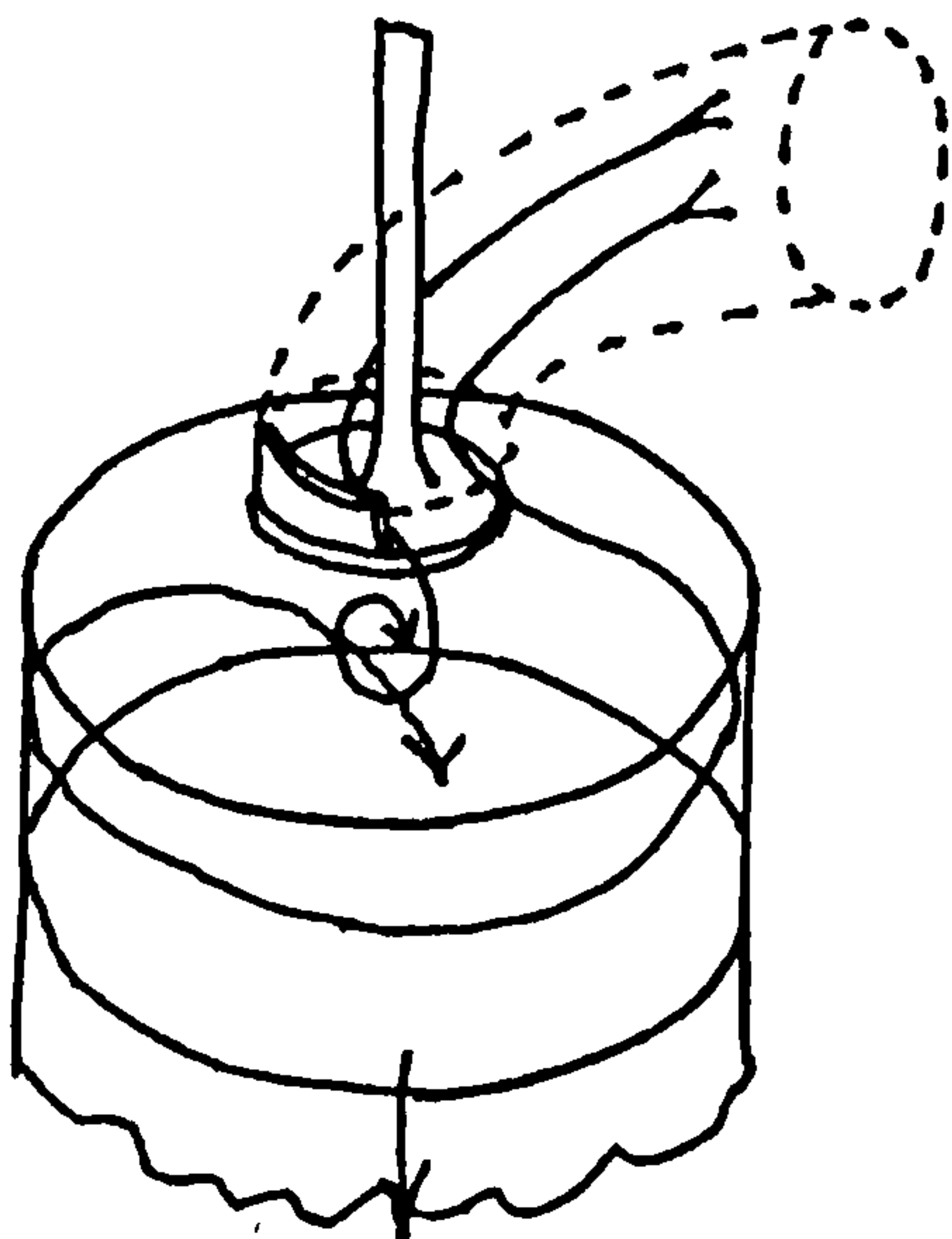
In this thesis attention is primarily focussed on fuel-air preparation and distribution prior to the combustion chamber and the assesment of its effect on the combustion processes and engine performance. However, the effect of combustion chamber generated

motion is also of great importance in the burning of lean mixtures as demonstrated in Chapter 2. In order to provide a general framework within which to assess the present studies the following sections in this chapter will mainly deal with some of the attempts made over recent years to improve our understanding of combustion processes in the spark-ignition engine.

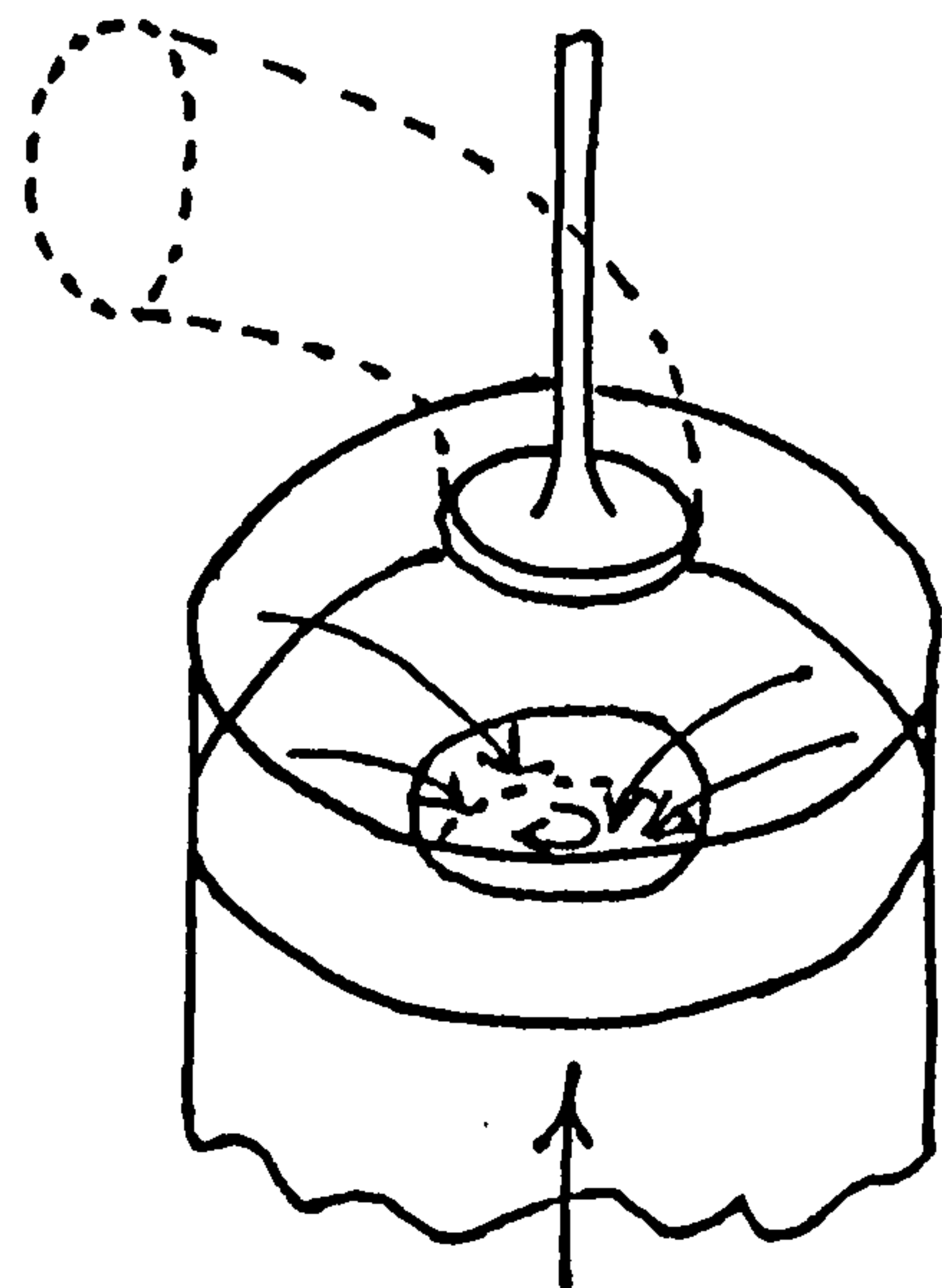
3.2 Generation of motion in the combustion chamber of a spark-ignition engine and its effect on combustion

3.2.1 Squish

Squish can be described as the inward motion of the cylinder charge created by piston movement near the end of the compression stroke (Fig. 3.2.a). Squish induced turbulence of the charge near TDC and also squish induced swirl motions have been variously described as a means of increasing the power output of the internal combustion engine by improving combustion rates.



(b) Swirl



(a) Squish

Fig. 3.2 Combustion chamber generated motion.

James (35, 36) studied the turbulent flow in the combustion chamber by comparing a disc-type chamber with a squish chamber, measuring velocity components. It was found that velocity fluctuations in the cylindrical chamber were lower than those in squish chamber. In other studies it was found that whilst the general characteristics of gas velocities are similar for most shapes of combustion chamber, the turbulence characteristics are mainly functions of inlet tract and combustion chamber geometry. Witze (37, 38) noted that the turbulence production by a squish volume would appear to be insignificant when compared to the compression stroke enhancement of intake-generated turbulence. He states that the turbulence structure is not homogeneous in the clearance volume but he also adds that it is not certain whether the degree of turbulence variation that exists is large enough to be significant to the mixing and flame propagation processes.

Nagayama et al (39) studied the effects of squish and swirl on combustion and emissions using a four cylinder 1.4 litre engine. They concluded that squish increased flame velocity and also increased torque and maximum pressure. Mayo (40) studied the effects of engine design parameters on combustion rate. In comparing a squish chamber to a disc chamber he found that the interaction between factors was large and although the squish design gave better results in some tests, inferior results were produced in others. However, he concluded that for the test series as a whole the incorporation of squish improved the combustion performance, although this seems to be significantly less than the improvements achievable through other modifications to engine design parameters, such as introduction of induction swirl, increase in charge velocity and spark gap location.

Lucas and Anton (41), in experiments using different types and degrees of squish configuration with both motored and fired engines, studied the effects of squish chambers on charge velocities, delay period, flame speed and turbulence intensity. They concluded that turbulence intensities are not significantly affected by squish whereas eddy size is increased by large squish areas and the eddy frequency also increases. However the ignition phase or delay period was found to improve with squish. On the other hand, the effect of squish on flame speed and mass burn rate were found to be small.

Overall, the effect of squish on combustion efficiency appears to be relatively small and as such squish on its own may not be considered to be the most effective means of improving combustion in the spark-ignition engine. However, factors such as intake velocities, intake swirl and spark plug location combined with squish can improve combustion since the effect of interactions between these parameters is normally large. Purely empirical observations are evidently of limited value in such highly non-linear conditions where detailed mechanisms are poorly understood.

3.2.2 Swirl

The large-scale rotational movement of cylinder charge is known as swirl. This movement is created mainly by the inlet port and the cylinder wall during the induction period (Fig. 3.2.b). There are several methods of producing swirl during the induction process. The two most commonly used methods are intake port configuration and shrouding or masking of the intake valve.

Swirl has been and continues to be used extensively in reciprocating engines to improve combustion rates by controlling and promoting better mixing of fuel and air before ignition. Swirl is a function of valve lift amongst others, and for this reason the evaluation of the average swirl and its effect over the valve lift is not possible without significant approximation. Ma (42) suggested the introduction of a dimensionless swirl coefficient, as defined by:

$$C_s = \frac{\text{Effective tangential velocity}}{\text{Resultant port velocity}}$$

The average swirl is then: $\frac{1}{\Theta} \int_0^{\Theta} C_s d\Theta$

where Θ is the valve opening period with respect to cam angle.

This definition of swirl coefficient enabled him to design quantitative experiments on the effect of swirl on combustion efficiency. His experiments were carried out on a disc-shaped combustion chamber, chosen to minimize squish and unaccounted turbulent motion. Over a range of port designs with different average swirl values he found the effect of swirl to be marginal and concluded that swirl on its own might not be the most effective means of inducing cylinder charge motion for promotion of lean burn in spark-ignition engines. However, on the other hand, Jones and Mackworth (43) found that controlled induction swirl in combination with a bowl-in-piston combustion chamber enabled significantly higher compression ratios and thermally efficient, spark-ignition engines which can operate on very low octane fuels. They also observed that these engines maintain efficient combustion and produce improved economy at lean mixtures when run on normal low octane fuels.

Nagayama et al (39), in a study of the effect of intensive swirl and squish on combustion and emissions in a spark-ignition engine, observed that swirl reached its maximum intensity in the first half of the intake stroke, and squish directly before the compression top dead centre. They also showed that an increase in swirl intensity reduced the ignition delay time and improved both the misfire limit and cycle-to-cycle fluctuation rate, and that these effects made lean operation possible, whereas an increase in squish extensity improved combustion rates at the main combustion stage and resulted in an increase in torque and maximum pressure. They concluded from their test results that the increased intensity of swirl, squish and swirl-squish combination is an effective way of improving combustion efficiency. In particular, they speculated that swirl was especially effective in the initial stage of combustion and swirl-squish combination was influential throughout the whole stage of combustion.

3.3 Mixture motion generated in the induction manifold

3.3.1 Fuel-air mixing and distribution prior to the combustion chamber

Spark-ignition engine operating characteristics in part depend upon the physical state of the fuel-air mixture inducted into the cylinder (21-26). The influence of mixture preparation is of particular importance when dealing with lean mixtures. Many authors (22, 25, 27) have shown that a fully vaporized, homogeneous fuel-air mixture provides good lean operating capability.

Fuel mixing and distribution in the spark-ignition engine can be altered by geometric changes immediately after fuel and air leave the carburettor, these changes can also alter the emission characteristics and lean mixture running of an engine. The induction of fuel and air into an automotive engine is an extremely complicated process. An engine has to provide good performance and fuel economy while still emitting very low concentrations of noxious gases. Performance, economy and emission levels have also to be met under variable engine speed and load conditions.

The expansion, vaporization, cooling and mixing of the fuel and air that takes place immediately after the carburettor and in the manifold runners is of particular importance, because it is in this region that most mixing and proportioning of the fuel and air is accomplished prior to burning. If a misproportion of fuel is sent down a runner, the ratio of fuel-air mixture for that cylinder will evidently deviate from design. On the other hand, if the correct measure of fuel flows in a given runner without satisfactory mixing within it, locally stratified rich and lean areas develop within the combustion chamber and variations in cycle-to-cycle fuel-air ratio may still occur.

3.3.2 Mixture quality

In principle it should be a straightforward matter to define mixture quality. Unfortunately, there is conflicting evidence as to what the optimum mixture state should be. For example, Matther

and McGill (21) have found that leaner burning operation was possible with "Bad" and "Wall-wetted" atomization than with "Good" atomization. They hypothesized that the "Bad" and "Wall-wetted" atomization resulted in some form of stratification and heterogeneous combustion. Peters and Quader (28) also suggested from their own test results that apparent heterogeneous mixtures of fuel and air give a leaner lean-misfire-limit than the homogeneously premixed charge case, and that some type of beneficial stratification must occur in the case of heterogeneous mixtures. Unfortunately, the data obtained in their study was not sufficient to delineate the configurations of the assumed stratifications. One possibility is that heterogeneous cases produce bulk charge stratification in the combustion chamber which place mixtures near the spark plug that were easier to ignite and burn. While this supposition could explain these results, it seems unlikely that bulk stratification could persist over a wide range of operating conditions.

There have also been many studies in the use of a premixed, preheated mixture. Jones and Gagliardy (29) have shown that the use of premixed, preheated mixtures improved the cylinder-to-cylinder distribution. They concluded that this was mainly due to the homogeneous nature of mixture in the manifold and as a result a leaner operation could be expected. Similar studies by Hughes and Golburn (30) have shown similar achievements and also some improvements in fuel economy and exhaust emission levels. In a later study, Hughes and Golburn (31) analysed the process of fuel vaporisation in air and have suggested that fully vapourised fuel in the inlet system may still not be homogeneous. On the other hand, studies by Beale and Hodgetts (32) suggested that mixture quality and its presentation had little or no significance except at low speeds and loads. However, their studies were based on single cylinder engines and at steady speeds and loads, and therefore they did not have a distribution problem comparable with multicylinder engines.

Improvements in mixture quality have also been achieved by better carburation. Wieatrak and Slawik (33) developed a carburettor device, the Szott carburettor, which produces smaller than normal droplets as well as generating micro-turbulence in the induction charge. They obtained improvements in fuel economy, faster combustion

rates, extension of the lean limit of operation and also improvements in volumetric efficiency. Similar views have been suggested by Kopa (34), whose studies of carburetion systems found that good fuel atomization and homogeneous mixing with air prior to admission into the inlet manifold eliminates power surging and extends the lean limit of operation.

Unfortunately, as demonstrated earlier, there seems to be conflicting evidence and opinions on the nature of the optimum mixture state. This is to a large extent due to the lack of detailed knowledge and understanding of flow structure in the induction manifold of spark-ignition engines. A better understanding of fuel droplet-air mixtures, fuel vapourization and combustion characteristics and their relevance to the spark ignition engine is also required if some of these conflicting reports are to be understood and explained. However, on balance, there seems to be more evidence to support the idea that better fuel-air mixing and atomization in the induction manifold (i.e. homogeneous mixtures of fuel-air) can aid the combustion processes.

3.3.3 Generation of mixture turbulence through throttling devices and vortex generators

As mentioned earlier geometric changes after the fuel and air leave the carburettor and before they reach the combustion chamber can have significant effects on mixture quality, emission characteristics and lean mixture running of spark-ignition engines. Over the years many attempts have been made to improve these properties by the use of alternative throttling devices and vortex generators.

Lucas, Brunt and Petrovic (44), in a study of the effects of vortex generation within the intake manifold and its contribution to the lean mixture running, made tests on a single-cylinder engine. A typical vortex generator used in their studies can be seen in Fig. 3.3. The vortex generators were positioned in the inlet pipe, the apex of the unit being directed towards the carburettor. They assumed that at the time of combustion, provided that the geometry of the vortex generator is near optimum, the resulting increase in level of turbulence would improve combustion efficiency. Their results

showed an extension of lean burn limits and an increase in measured flame speed by the use of vortex generators. They concluded that a low blockage ratio vortex generator was sufficient at high engine speed and high load ranges, whilst a high blockage ratio vortex generator was necessary at low engine speeds and low loads. Hence, they suggested that a variable blockage ratio vortex generator could be a desirable alternative to the conventional throttle (i.e. butterfly valve) to cater for the full engine load and speed range.

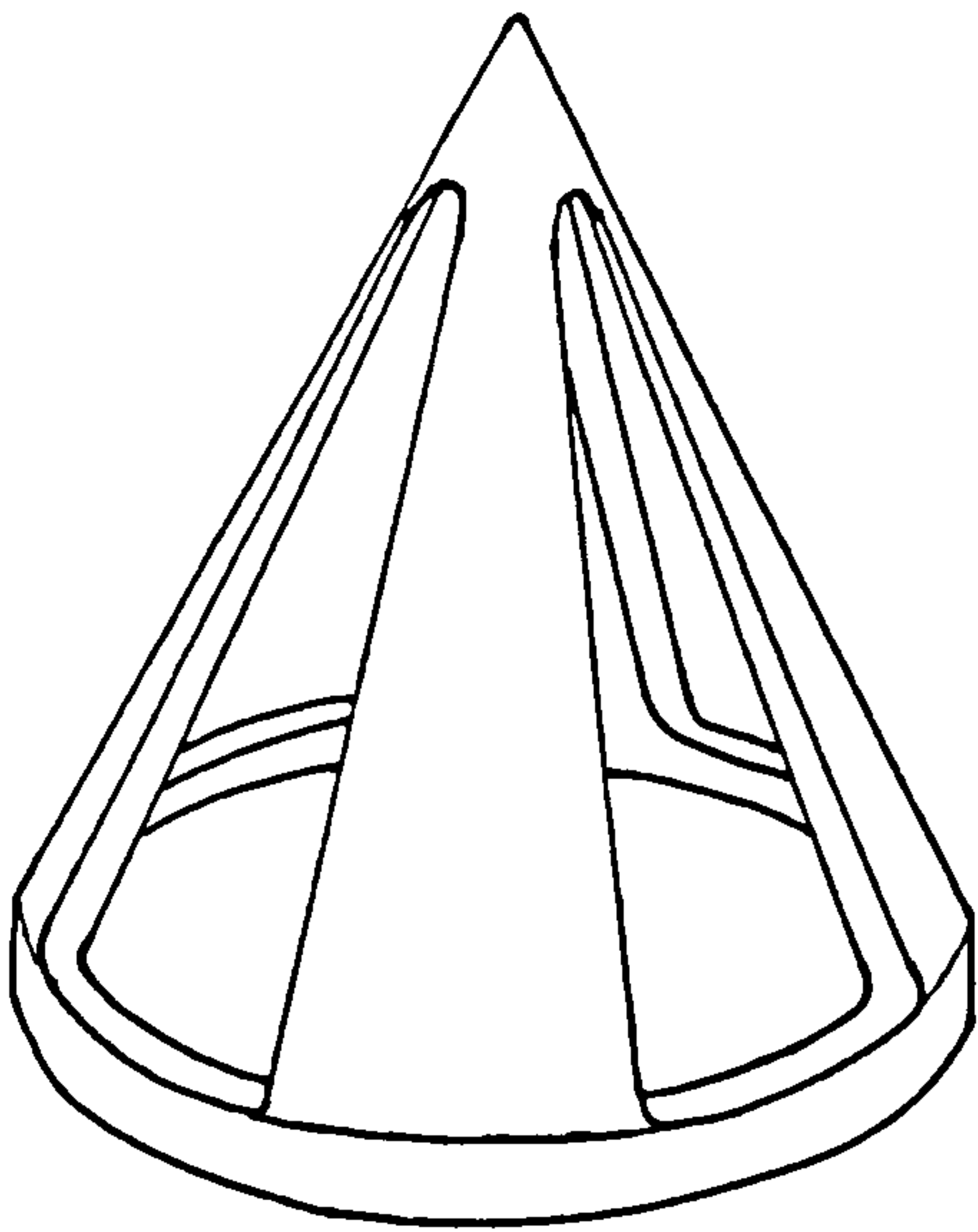


Fig. 3.3 A typical vortex generator.

Inlet valve throttling may not be classified strictly as a means of generating mixture motion in the induction manifold of an engine, but at the same time it has been shown to be capable of improving combustion by increasing mixture flow velocity through the valve port. This is achieved by variation of inlet valve full lift in place of a conventional throttle device (i.e. butterfly valve). Stivender (45), in a study of a spark ignition concept employing a sonic throttling intake valve, demonstrated that the rapid turbulent combustion process so produced permitted extremely lean part-load operation. He concluded that the improvement in combustion efficiency was the result of increased intensity of the small scale turbulence produced

during throttling by means of the intake valve. In a later study Beals and Hodgetts (46), in an evaluation of valve throttling compared with normal throttling, examined the effects of valve throttling on fuel consumption and gas emissions of carbon monoxide, nitric oxide and hydrocarbons. They concluded that the extra mixing and turbulence which is generated by the high velocity through the inlet valve aperture of a valve throttling engine is responsible for the extension of the lean limit of combustion and low emissions of carbon monoxide, at part load and idling conditions. Overall, the variable lift valve, as a throttling device, has been shown to be an effective means of providing improvements in cylinder turbulence at part-load and closed-throttle conditions. This is especially important as mixture quality is more significant at low engine speeds and part-throttle operation in the efficiency of combustion.

Based on the general observation that the higher the mean flow velocity and the more abrupt the discontinuity the finer and more intense the transformation of mean flow kinetic energy to turbulence, Ma (47) designed a shock valve generator (Fig. 3.4) as a means of throttling control in place of the conventional butterfly valve.

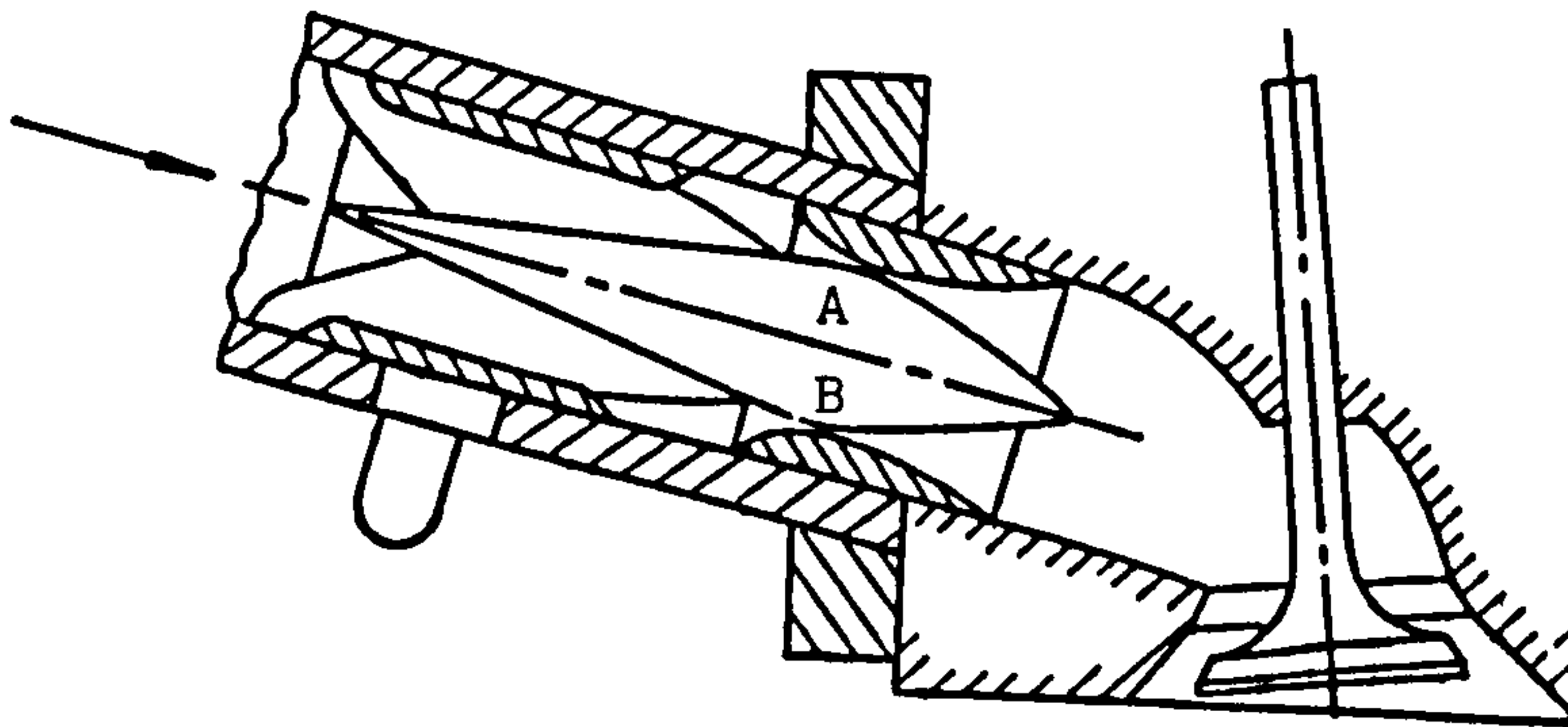


Fig. 3.4 Turbulence generator

Variation of engine mass flow in his design was achieved by the axial movement of a plunger to vary the throat area. Sonic velocity is supposedly reached during intake at the annulus throat section AB and a shock wave develops further downstream, depending on the prevailing manifold vacuum. From the tests he made on this design, in comparison with conventional throttling devices in spark-ignition engines, he concluded that better rates of combustion and lean limits are achievable with the alternative throttling device, and that small scale, high intensity turbulence generated by such devices inside the inlet manifold can result in the promotion of lean burn.

3.4 The variable ramp throttling device

The variable ramp throttling device is of particular interest. It is the purpose of the present study to examine the effects of this particular design and to compare its performance with the conventional butterfly valve on the induction flow process and cylinder charge motion. The principal tool in this examination will, however, differ significantly from the several studies described to date, in that it will seek to make a detailed analysis of the flow using computer modelling and simulation. It is important, however, to establish the practical base for the study first.

3.4.1 Description of design

In 1981-82 a study by Abbas (48) at the Cranfield Institute of Technology produced a design for an inlet throttling device. This particular device consisted of a convergent-divergent passage formed between a fixed side and a rotating flap, as illustrated in Fig. 3.5. It replaced the conventional butterfly valve as a means of throttle control, by using the rotating flap to change the nozzle throat area.

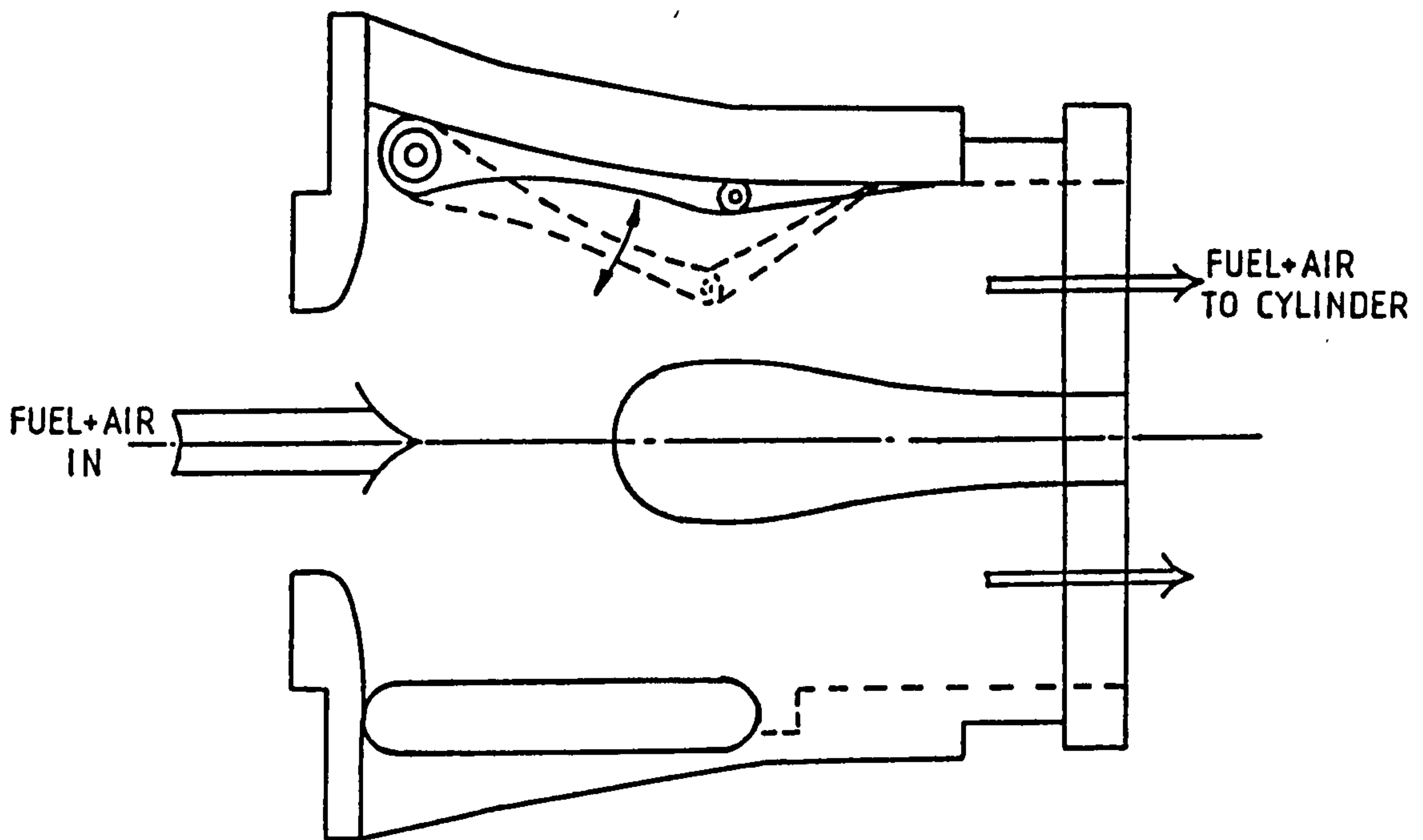


Fig. 3.5 Variable ramp or turbulence generating throttling device

3.4.2 Principle of operation

The principle behind the design was again based on the idea of increasing the mixture velocity at the inlet port and through the inlet valve seat. In addition, it was suggested that when the throttling device was used as a throttle control at part loads sonic velocities might be reached during intake at the device's throat and a shock wave might occur downstream, depending on the prevailing manifold conditions. As a result of this, it was argued that intensive micro-turbulence might be created as well as better fuel atomization, which in turn could aid combustion efficiency and the lean combustion limit by the introduction of a homogeneous air-fuel mixture into the combustion chamber.

3.4.3 Tests and results

Tests were carried out on a 1500 cc four-cylinder engine, under a variety of engine speed and load conditions. From his test

results Abbas (48) suggested that under wide-open throttle conditions the device increased the charge velocity resulting in better combustion. He attributed this to the possibility of better mixture preparation at the inlet port resulting from increased turbulence. His report indicated that improvements in power and fuel economy were especially significant with the variable ramp throttling device at wide-open setting and fuel-air ratio on the lean side of the stoichiometric ratio. Strangely enough, he reported no appreciable effect on combustion at part loads and with rich mixtures. He suggested that this might be the result of leakages through the clearances between the flaps and the main body.

In a subsequent study, Abdulla (49) undertook a similar investigation to that of Abbas (48) in order to gain a better understanding of the effects of the turbulent generating throttling device on combustion. His primary objectives were to fully check and evaluate the performance of the earlier throttling device by Abbas (48), to confirm the reported gains of the previous study and to investigate methods to improve the part-load performance of the device.

After extensive tests carried out by Abdulla (49) to compare the performance of the throttling device with that of the butterfly valve, he reported the following improvements:

- An appreciable improvement in BSFC at all engine speeds.
- An extension of lean limit burning, and significant gain in power and fuel economy.
- An overall improvement in engine performance, both at full and part loads. The improvement in part-load performance of the device was achieved by elimination of the leakages which arose due to the clearances between the flaps and the main body of the throttling device, when tested previously by Abbas (48).

Overall, both studies substantiated the claims for the superiority of the variable ramp throttling device over the conventional butterfly valve. Both authors attributed the apparent improvements in engine performance to increases in turbulence level resulting from increases in inlet velocity past the valve (i.e. higher rates of combustion), better fuel mixing and atomization in the inlet manifold and better volumetric efficiency due to improved mass flow.

3.4.4 Computer simulation studies

Whilst these were plausible hypotheses, the lack of detailed knowledge of the fluid flow in the induction tract and its effect on cylinder charge motion made it impossible for the authors (48, 49) to identify the mechanisms clearly and hence to explain the exact cause of their reported performance improvements. In common with many earlier studies their observations were to a large extent simply speculative.

Responding to the need for more detailed knowledge on the induction tract of the system, Sanatian (50) undertook an analytical study at Cranfield with the objective of simulating the detailed flow through the induction tract with the aid of computer models. The general purpose code PHOENICS, described briefly in Chapter 6, was used to simulate a series of basic flows relevant to the induction tract. In addition, many of the built-in functions were validated using simple idealised flows. Preliminary two-dimensional comparisons were made between the computer model of the variable ramp throttling device, simulated simply as a convergent-divergent duct and butterfly valve under steady flow conditions. Increases in turbulent kinetic energy were reported downstream of the device, roughly in the position where the combustion chamber might be situated in the case of the alternative throttling device, again suggesting a better mixing of charge and higher turbulence intensities. However, the models used in the study (50) were relatively crude and did not satisfactorily represent either duct geometry or the flow through the butterfly and variable ramp systems. The results and observations made during the study (50) did however indicate a promising future in the application of PHOENICS for flow simulation in internal combustion engines, and encouraged these further studies.

3.5 Investigative methods and approaches

The fluid motion within the cylinder and induction system of a piston engine has a major influence on the performance of the engine. In order to predict the mixture quality and distribution one must understand and be able to predict the turbulent mixing processes. Combustion analysis thus requires knowledge of turbulence structure and the wide range of prevailing turbulence scales. Confinement further complicates the turbulent processes. For example, the flow in wall boundary layers controls the heat transfer and quenching. Since pollutants are formed in varying amounts in different regions in the cylinder, analysis of the exhaust emissions also requires the ability to predict which fluid elements will be purged during the exhaust and which will remain. Hence, a complete predictive capability requires detailed knowledge about the general circulation in the induction system, the cylinder, embracing both turbulence and the interaction with boundaries.

The traditional approaches, modest extrapolation based on past experience, trial and error experimentation and semi-empirical analysis of the lumped parameter variety have shown themselves to be of limited value. Although they proved to be successful when allowed to proceed at a fairly gradual pace, when pressed more vigorously by the need for radical changes to meet new requirements, the extrapolations have become too large, the required experiments too numerous and the analytical techniques too empirically-based to allow further extension. More fundamentally, it has been recognized that these inadequacies stem primarily from the fact that despite the long history of development of the reciprocating internal combustion engine, very little quantitative analytical information is available about structure of the flow, mixing and combustion processes within the combustion chambers, and the important governing parameters and their effects.

Computational fluid mechanics has been a distinctive and active topic for over two decades, and has achieved some substantial successes during this time. Many complex three-dimensional flow phenomena are now being successfully simulated by computer and important features of turbulent flows have been satisfactorily reproduced by solutions of modelled turbulent flow equations. This development has been greatly

aided by the substantial reduction in costs and increase in capabilities of computer hardware. There has also been progress on the mathematical side of the development of efficient and accurate algorithms, but arguably on a rather smaller scale. It can not yet be said, however, that computer simulation of fluid flow has become the standard recourse of the automotive engineering industry in seeking improved designs. This attitude is changing. Computer simulations of internal combustion engine cycles are becoming more prized because of the contribution they make in flexible and responsive design studies, in predicting trends, as diagnostic tools, in providing more data than are normally obtainable from experiments, and in helping to understand the complex interactive processes that occur. On the other hand, computational fluid mechanics is still in a development stage in many non-aeronautical applications, and there are still important steps to be taken before it replaces development trials, particularly for lower cost capital items such as some of those in the automotive industry. A more detailed discussion of computational modelling of fluid flow will be made in Chapter 6.

The computer analysis provides a representation of the problem at hand constrained by the implications of a mathematical model. The experimental investigation, by contrast, can in principle observe the reality itself. The most reliable information about a physical process should therefore be given by actual measurements. Point flow measurement techniques have therefore been used for a much longer period of time as a diagnostic, and are still being used to a greater extent than computational techniques to provide insight into the details of flow processes in engines. There are however significant drawbacks with these methods. In the engine system there is, for example, a serious problem with basic definitions of turbulence, since the flow is not steady. The fluctuating component of a variable can not be simply defined as the departure from its time-averaged value, as one does in steady flow. In periodic flows the concept of an ensemble- or phase-average has proved crucial. The phase-average is defined to be the average of values at a given phase in the basic cycle over many such cycles. The phase-average velocity, for example, would be the average over a large number of measurement cycles taken

at the same crank angle. The difference between the instantaneous velocity and the phase-average velocity is then defined as the fluctuating component of the flow field. Under this definition the turbulence is the departure from the average over many realizations, but will not necessarily simply relate to the random fluctuations in space or time that exist in any single realization. For example, a laminar flow with large cycle-to-cycle variations will appear to exhibit a high degree of turbulence. In such a case turbulence defined in this way can have little relation to mixing processes occurring in each single realization. The inhomogeneity of engine turbulence further complicates engine turbulence measurements. The large-scale motions are quite anisotropic in structure and very dependent upon system geometry. The smaller scales, on the other hand, are locally isotropic, but their magnitude may vary with position. These spatial variations are especially pronounced during the intake process, and this makes characterisation of the process by a global measured value of rms turbulence velocity, or the turbulent length scale during intake or similar processes quite meaningless. In addition, the hostile and highly non-linear environment of the combustion chamber poses formidable problems of accessibility for the instrumentation. There still remain a need to distinguish engine turbulence in a more meaningful way, and a great need for good experimental data from which the physically important characteristics of engine turbulence can be determined as functions of position and phase in the engine cycle. These data are particularly useful in guiding the development of flow models and are essential to their validation. In the meanwhile less detailed experiments, designed to take into account the limitations of measuring techniques, can provide valuable field, rather than localised, information for development of complex computational models of flow in internal combustion engines.

It is generally accepted that "seeing is believing", and hence to observe the flow in relation to its constraining boundaries yields valuable information to verify a specific design and provide the basis for design modification. Flow field visualization has been successfully used to provide such information for much of this century. Such methods are mainly of importance in providing a qualitative picture of the flow phenomenon, albeit in model configurations. In

addition, attempts have been made over the years to use flow visualization for quantitative information, but with much less success. The methods usually depend either on the reflection or scattering of light by small solid or liquid particles introduced into the stream, or on the natural density changes of a compressible fluid, or on the reflection of light by filaments or particles of fluid of a different refractive index introduced into the flow. The need for partially or totally transparent models of the system under investigation limits the scope of the study in systems such as internal combustion engines. The technique will be described in greater detail in Chapter 4 in relation to the present investigation of induction systems.

In the context of flow simulation in internal combustion engines, where complex, transient fluid flows are involved, steady-flow studies of such systems using point flow measurement and flow field visualization avoid some of the ambiguities mentioned and the interpretive problems involved with transient flows. They can then provide valuable information for the progress, development and validation of computer models of this complex flow. This is the general approach adopted for this present study. A more detailed discussion on each of these techniques will be made in subsequent chapters.

3.6 Concluding remarks

The literature review reveals that there are conflicting opinions as to both the optimum mixture and the most effective induction manifold geometry for its generation. This state of affairs arises in part from the lack of detailed knowledge of the different flow processes in the induction system and the combustion chamber of the spark-ignition engine, and their individual or combined effect on different stages of the combustion process. However, a number of useful conclusions can be drawn:

- The separate introduction of squish or swirl into the combustion chamber have been shown to be of limited value in promoting mixture motion, mixture turbulence, and hence improvements in combustion efficiency. However, when used together their interaction may promote

better combustion efficiency overall, by improving combustion rates at different stages of the combustion process. The nature and extent of these improvements can further depend on combustion chamber shape and spark igniter position. A more specific statement on the detailed effects of such cylinder motions on combustion requires a much better understanding of the interactive flow phenomenon at different stages through the engine cycle than is available at the present time.

- The attainment of homogeneity in the fuel-air mixture in the induction manifold is generally understood to aid lean mixture running of spark-ignition engines, by improving combustion rates.

- The generation of increased turbulence, with the aid of alternative throttling devices can not only improve mixture preparation (i.e. homogeneity and distribution), but in addition can improve combustion rates by introducing or maintaining otherwise absent cylinder charge motions.

- The ability to obtain detailed information of a multi-dimensional and multi-phase nature in hostile and geometrically complex environments such as the ones prevailing in internal combustion engines by computer simulation and modelling appears to be the most effective means of progressing further our understanding of processes and the further development of engine design.

CHAPTER 4

AERODYNAMIC FLOW VISUALIZATION STUDIES OF INDUCTION FLOWS

4.1 Introduction

The extensive research carried out on the fundamental, physical and chemical processes of combustion in spark-ignition engines has not always been satisfactorily linked with their practical operation. As requirements become more demanding, further progress and development in the field of engine design must be related more satisfactorily to the fundamental fluid flow phenomena if the full benefit of combustion and fluid flow research is to be achieved. Given the idealized nature of most of this research, an intermediate stage is needed between the two extremes of basic research and applied combustion in practical systems. The component processes in the engine need to be examined individually in relation to their role in the integrated process. This is made possible by considering the processes of fluid motion, combustion and air flow individually, but in circumstances appropriate to their behaviour in the actual engine cycle. Co-ordinating the component studies in this way permits significant improvement in our understanding of overall engine performance.

In the present study, the influences of two different throttling systems on induction flow processes in relation to flow mixing and distribution are compared. Flow field visualization by water analogy can provide valuable qualitative information about the structure and main features of flow in such systems. It also provides a means by which computational models of the flow can be partly validated. However, a thorough study of the flow processes is very difficult to achieve in the presence of combustion and transient effects, whereas cold steady flows can be investigated comparatively routinely. In the present study the induction flow processes -under cold and steady conditions- are assumed to simulate the gross features of flow such as recirculations and flow separations closely. Initial comparisons between computer simulated results and flow visualization results

will also be made on steady flow basis.

For the direct visualization of cold flows it is generally feasible to construct completely transparent models of the engine systems under study without excessive cost or difficulty. Models of this sort can be constructed to any convenient scale, provided that geometric similarity is retained in the essential features, and provided that due regard is paid to satisfying the necessary dynamic similarity criteria. This will be discussed in more detail later in this chapter. In addition to complete three-dimensional models, simpler representations are of value for preliminary analysis where the flow is, for example, predominantly of two-dimensional nature. In the present investigation, however, the study will be of the fully-dimensional flow.

Flow-field visualization study of internal combustion engine systems can be regarded in the broadest sense as including any direct visualization in which the motion of the fluid is rendered visible by some type of passive tracer and can be observed by eye, or, if the motion can not adequately be studied directly, can be recorded photographically. The resulting photograph can then be interpreted qualitatively or, less commonly, measured by suitable instruments and perhaps simulated by computer models.

The experimental work reported in this chapter is of the direct visualization type. The flow field in full-scale models of the complete induction system, comprising simplified representations of the induction tract, throttling device, inlet valve and cylinder are studied by the means of steady-state water analogy tests. The primary function of these tests was to provide quantitative information on the main features of the flows and, in particular, on the extent of mixing, recirculation and turbulence distribution, through the variable geometry ramp and butterfly valve systems. This information is used both diagnostically for direct comparisons of mixing and turbulent processes between the two alternative systems and for validation and assessment of computational modelling results in the later stages of the study (i.e. Chapter 6).

4.2 Choice of working fluid and similarity conditions

4.2.1 Choice of working fluid

As has already been mentioned in the introduction, the study of flow under cold, or more correctly isothermal, conditions allows some freedom in the choice of the fluid used in the simulation.

Air is an obvious medium for this type of work since the same fluid is employed in the actual engine. However, it is not ideal from other points of view and it is advantageous to substitute water for air. We must observe immediately that there is a fundamental difference between the two fluids in respect of their compressibility, but this does not invalidate the use of water in the present study since the Mach Number of the flows investigated are everywhere small.

Fig. 4.1 is a plot of approximate Mach Number at the valve throat during the intake process of a typical production engine. These values were calculated using incompressible flow equations with actual valve lift profiles. However, the reverse flow through the intake valve that normally occurs during the early part of induction was neglected. It is clear that during intake Mach Numbers at the valve throat, the point of maximum velocity, are generally higher than 0.3 for high engine speeds. Since this is the commonly accepted threshold for neglecting compressibility effects, the water model can not be a valid analogy of the actual flow fields over the complete range of engine speeds. In the present studies, however, steady flow simulations are based on relatively low engine speeds (say, 2500 rpm) and high valve lifts. Consequently the Mach Number of the flows investigated are everywhere smaller than 0.3.

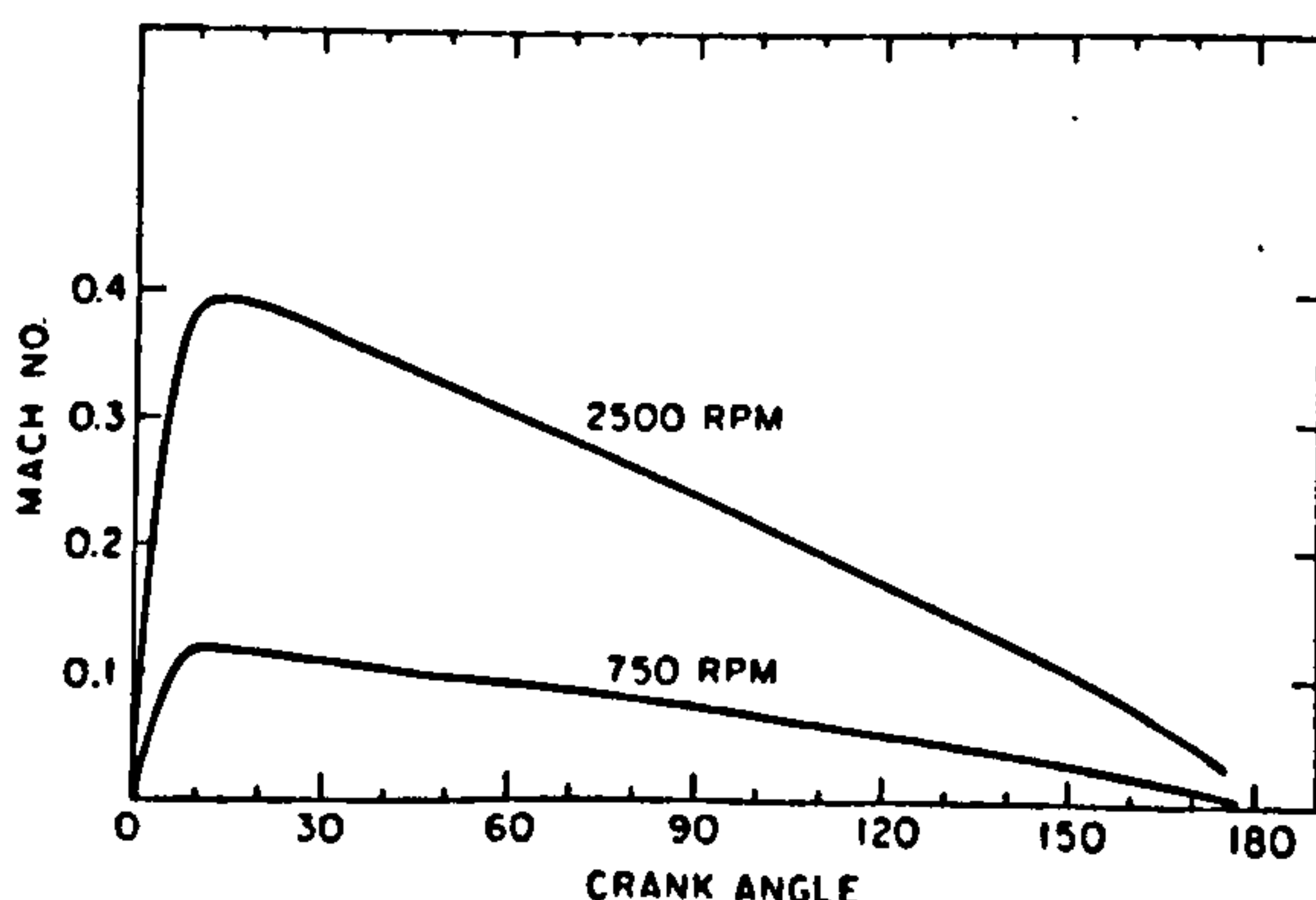


Fig. 4.1 Mach Number at valve throat during the intake process. (From Ref. 51)

In order to appreciate the merits of water over air as a working fluid one needs to look at the techniques available for the visualization of air flows. This is possible in three-dimensional transparent models using light scattered from smoke particles or some other particulate tracer. Smoke techniques have been used widely in general aerodynamic studies, but they are generally only of value for the study of local conditions marked by a smoke filament, and will not show more extensive flow patterns. The filament is rapidly dispersed in highly turbulent flows and the marker loses the necessary contrast. In addition, many chemical smokes cause at best inconvenience because of their unpleasant odour and irritant or corrosive qualities. Discrete particles of a suitable size, however, can be used to trace the air flow, provided that they can be observed satisfactorily by scattered light. Unfortunately the tracers available are mostly very much denser than air, and the accuracy with which they follow an air stream is an inverse function of their size. For example, in a work by Nicholson and Fields (52) aluminium powder was used for visualization of an air stream approaching a flame stabilizer. The powder appears to have followed a normal population type of distribution, with the greatest number of particles in the lowest sizes, and more than 50% by number were believed to be below 3μ . Theoretical predictions of the flow characteristics were used to confirm the validity of the use of these traces, but the conditions of flow in this case were of a very simple nature, with only mild accelerations and changes of direction. Under more exacting conditions the larger particles present would undoubtedly have contributed more significant errors. This results in rather poor tracing in the sort of complex flows which occur in the present systems. An additional difficulty sometimes experienced with tracers in air is the settling out of the particles in comparatively stagnant regions and on walls. Moisture and electrical charge separation can both cause the precipitation and adhesion of small particles to the surfaces of the model, with a consequent decrease in clarity.

On the other hand, assuming for the moment that a measure of dynamic similarity is maintained, the use of water in a given size of model permits substantially lower velocities than with air. This makes it very much easier for the motion to be observed by eye

or to be photographed. It is also possible to introduce particulate which have substantially the same density as water. This permits the use of tracers of a convenient size from an optical scattering point of view whilst retaining flow fidelity, and consequently leads to a more rapid and more exact appreciation of the motion of the fluid. When the condition of the water is properly maintained, the optical clarity through a system in which the bounding surfaces are of glass or Perspex is always excellent. The self-cleaning action of a water simulation system is very much superior to that of air. The major disadvantage of water is that the higher pressure, which typically occurs for a given size of model at a given Reynolds number may result in the inconvenience of leakage from the test equipment.

4.2.2 Flow visualization studies with water

For isothermal flow studies, water models have significant advantages over air models as indicated. They have therefore been widely used for the investigation of many aerodynamic problems, including such diverse applications as the flow around aerofoils, through industrial furnaces, through the combustion chamber of gas turbines and in reciprocating engines.

Flow visualization with water in a closed system or tunnel can be accomplished by similar techniques to those used with air. For example, streamers or tufts can be used to show the direction of the flow, and dyes can be injected to indicate local mixing patterns in the same way that smoke is used with air. But water has the particular advantage over air that it is possible to provide discrete tracers of a similar density, so that the technique can perhaps be used more confidently. Variations of the discrete tracer technique will permit the study of almost any flow visualization problem to which a water model is applicable. Probably the only important exception to this is the study of the behaviour of a jet of fluid discharging into a similar fluid, which can be studied more appropriately by means of dyes in one or both of the mixing streams. But dye techniques are rather inconvenient in the models involved in the present study, because once complete mixing has occurred the flow is no longer visible.

Tracers for flow visualization in water should ideally comply with five principal requirements, which are as following:

- They should remain as discrete bodies in order for markers to retain the necessary contrast.
- They should have similar density to that of the fluid.
- They should be of spherical shape in order to flow steadily with the fluid under all flow conditions and in order to exhibit consistent optical properties when illuminated. This, however, may be more appropriate in unsteady flow cases.
- They should be large enough to permit the observation and photography of individual tracers so that the details of local flow can be distinguished. But they must also be of a lower order of size than the flow details which are to be studied.
- They should exhibit high proportional reflection or re-radiation of light in directions normal to the incident beam.

The first of these requirements is met in general by insoluble solids and, to some extent, by some liquids when introduced as small droplets. The density requirement is however of varying importance depending on the complexity of flow and on the accelerations which occur. In general, if tracers of a density appreciably different from the fluid are used, it is necessary to use very small particles for following the flow, as in the case of solid tracers in air.

Introduction of air bubbles in water has been found to provide a simple means of fulfilling most of the principal requirements laid previously.

The major requirement which can not be completely satisfied is the density. However, the difference between the density of the air bubbles and water can be minimized by the use of smaller air bubbles. In addition, unlike a solid tracer particle, a gas bubble can change its shape during the motion with the fluid flow, and as a consequence the drag coefficient of this gaseous tracer particle is not only a function of velocity difference between the fluid and the tracer but also a function of deforming forces acting on the particle. A quantitative analysis of the bubble motion in the fluid is therefore very complicated, and it must also account for the rise rate of the bubble due to buoyancy. When the flow patterns are highly

turbulent, however, and the flow paths of individual particles deviate from the mean flow path the question of density difference and buoyancy effects is of secondary importance in tracing of gross features of the flow. This means that gaseous tracers, of which the most convenient is air, can be introduced for tracing out flow patterns in the present studies. This is the technique used in the present study, for in addition to the advantages discussed, there is a pool of experience available in Cranfield Institute of Technology in the application of this particular approach in flow visualization. A satisfactory pattern is usually traced out using bubble diameters of the order of 0.1 mm for water velocities above 1 m/s. Below this limit, however, the buoyancy effects are present, and at high water velocities the diffusion of the air into very small bubbles can become so extensive as to make observation difficult.

4.2.3 Dynamic similarity condition

The classical work of Osborne Reynolds which resulted in the concept of the Reynolds number as a dimensionless similarity parameter forms the foundation of flow and model studies. The Reynolds number is conveniently expressed as the ratio of the inertia forces to the viscous forces present in a moving fluid. Providing the above ratio is the same for both the model and the practical system, and the fluids conform to the Newtonian definition of viscosity, then no matter what the fluids are, or the size of model, the flow patterns will be substantially similar. This means that scale models can be made of systems whose actual dimensions, whether large or small, prohibit the use of full-size models. Also fluid media different from those in the actual equipment may be used. In the present study the actual dimensions of the system were such that full-size models could be used. However, the fluid media can be usefully changed from air to water.

Before embarking on a programme of flow investigation, consideration should be given to the several variables involved. In the present study gravitational forces and compressibility are assumed to be negligible. The principal criterion of similarity of flow pattern

is then Reynolds number, Re , which is expressed in the non-dimensional form:

$$Re = \rho v d / \mu$$

where v represents the mean velocity of the fluid,
 d represents a characteristic linear dimension associated with v ,
 ρ represents the density of the fluid,
 μ represents the absolute viscosity of the fluid.

Since $Q = Av$, where Q represents the fluid volumetric flow rate through area A into the system, then:

$$Re = \frac{\rho Q d}{\mu A}$$

Hence for dynamic similarity

$$\frac{\rho Q d}{\mu A} \text{ for air} = \frac{\rho Q d}{\mu A} \text{ for water}$$

or, for common physical scale,

$$Q_{\text{water}} = Q_{\text{air}} v_{\text{water}} / v_{\text{air}}$$

This would mean that the rate of volumetric flow for water should be approximately one tenth of that for air, in order to maintain dynamic similarity.

On the other hand, when the Reynolds number is large enough throughout the system, it is not necessary to insist on its exact identity but solely on a large enough value in the models to ensure that drag is independent of this parameter. Indeed, Gerrard and McAreaney (53) state as a generalization that "the operation range of a (gas-

turbine) combustion chamber is such that a very wide variation of the Reynolds number will bring about no significant change in the flow pattern". Young and Millar (54) have also expressed this view with regard to flow in ducting systems generally. It is necessary, however, to moderate this statement with respect to present systems. Within the main flow pattern of the induction manifolds, the statement is on the whole quite fair, since the Reynolds numbers there under all practical conditions will be of the order of tens of thousands, and provided the value of the model is then maintained above about 10^4 no significant change of flow pattern will be observed.

4.3 Equipment and set up

4.3.1 Transparent models of the induction and throttling systems

Flow visualization techniques naturally require totally or partially transparent models of the systems under investigation. In the present study, totally transparent models, favoured for the flexibility and range of viewing options they provide, were designed and manufactured. Geometric dimensions of the induction tract, throttling devices (i.e. butterfly valve and variable geometry ramp), and inlet valve and cylinder were reproduced at full-scale in Perspex.

Each manufactured Perspex model consists of two main sections: the induction tract and throttling device shown in Fig. 4.2, and inlet valve and cylinder shown in Fig. 4.3.

The inlet valve and cylinder sections were designed to fit either the butterfly valve system or the variable ramp system. In the Perspex model of the butterfly valve system (Fig. 4.2.a) the butterfly valve is incorporated in the inlet pipe to the system and can be set to any desired angle. Immediately after the valve, the flow is divided by a 'trouser leg' pipe representing the inlet pipes to two cylinders. One of the induction pipes was blocked (Fig. 4.2.a) in order to represent flow stagnation through it, during the induction stroke for the other manifold.

In the case of the variable ramp system (Fig. 4.2.b) throttling

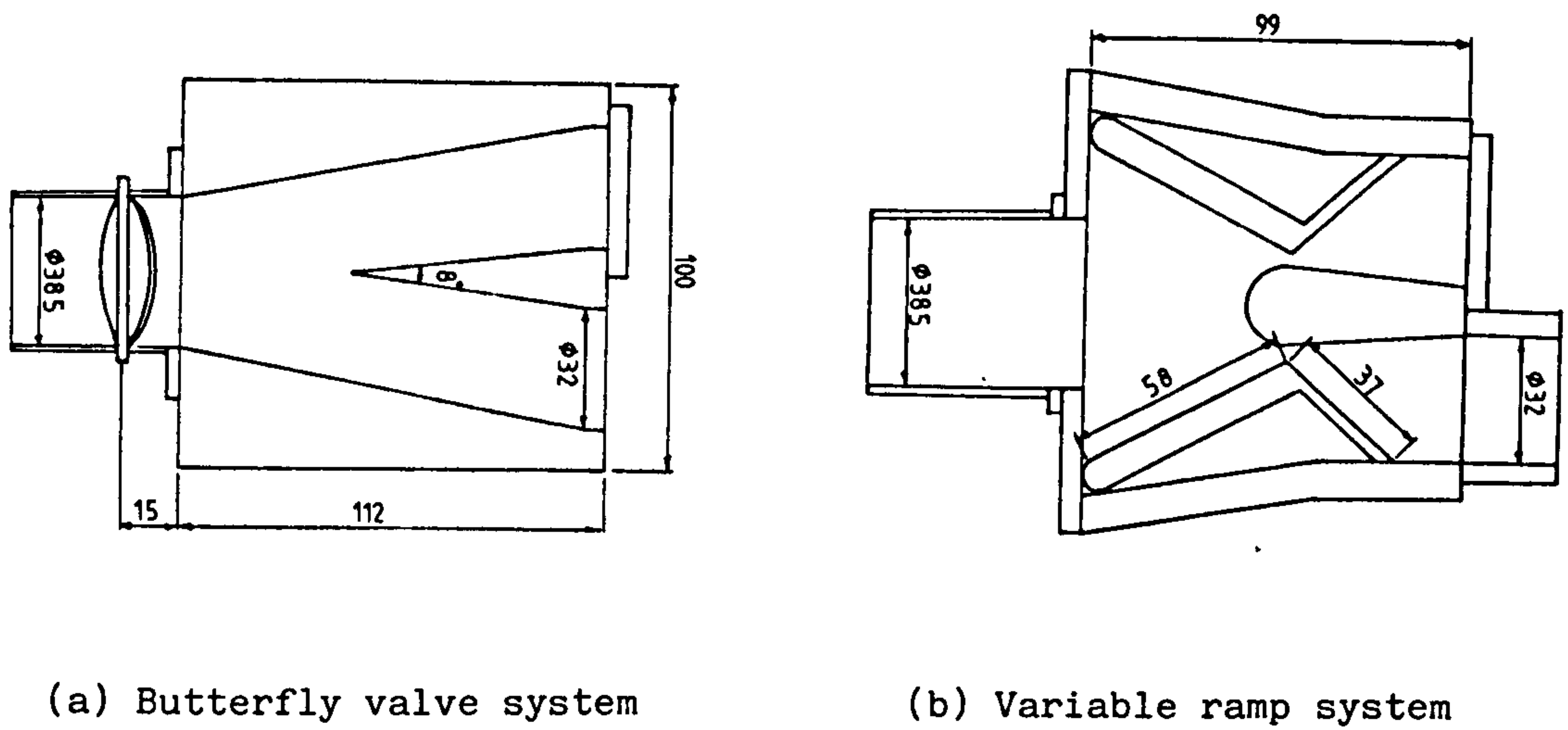


Fig. 4.2 Perspex models of the throttling systems.

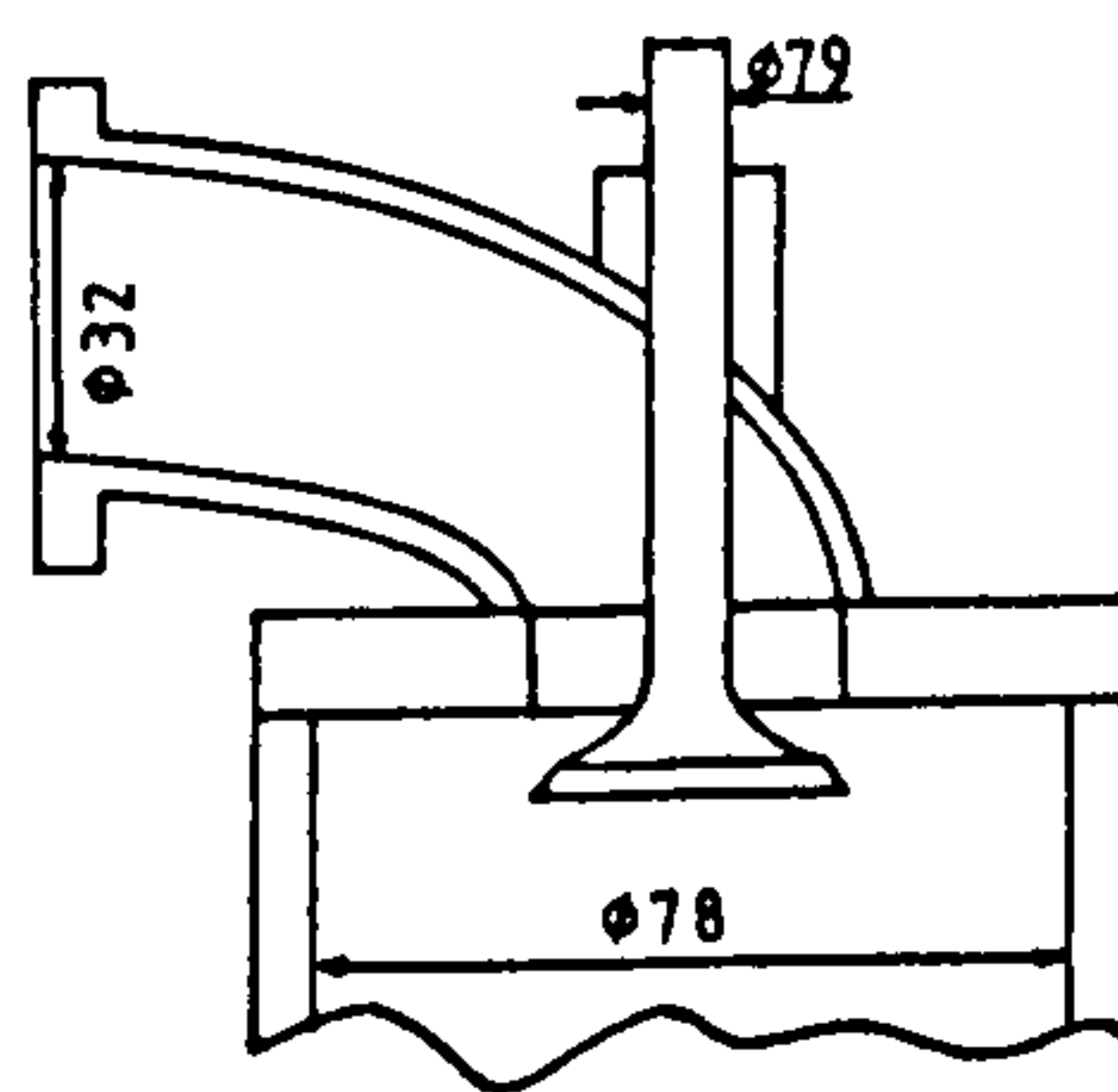
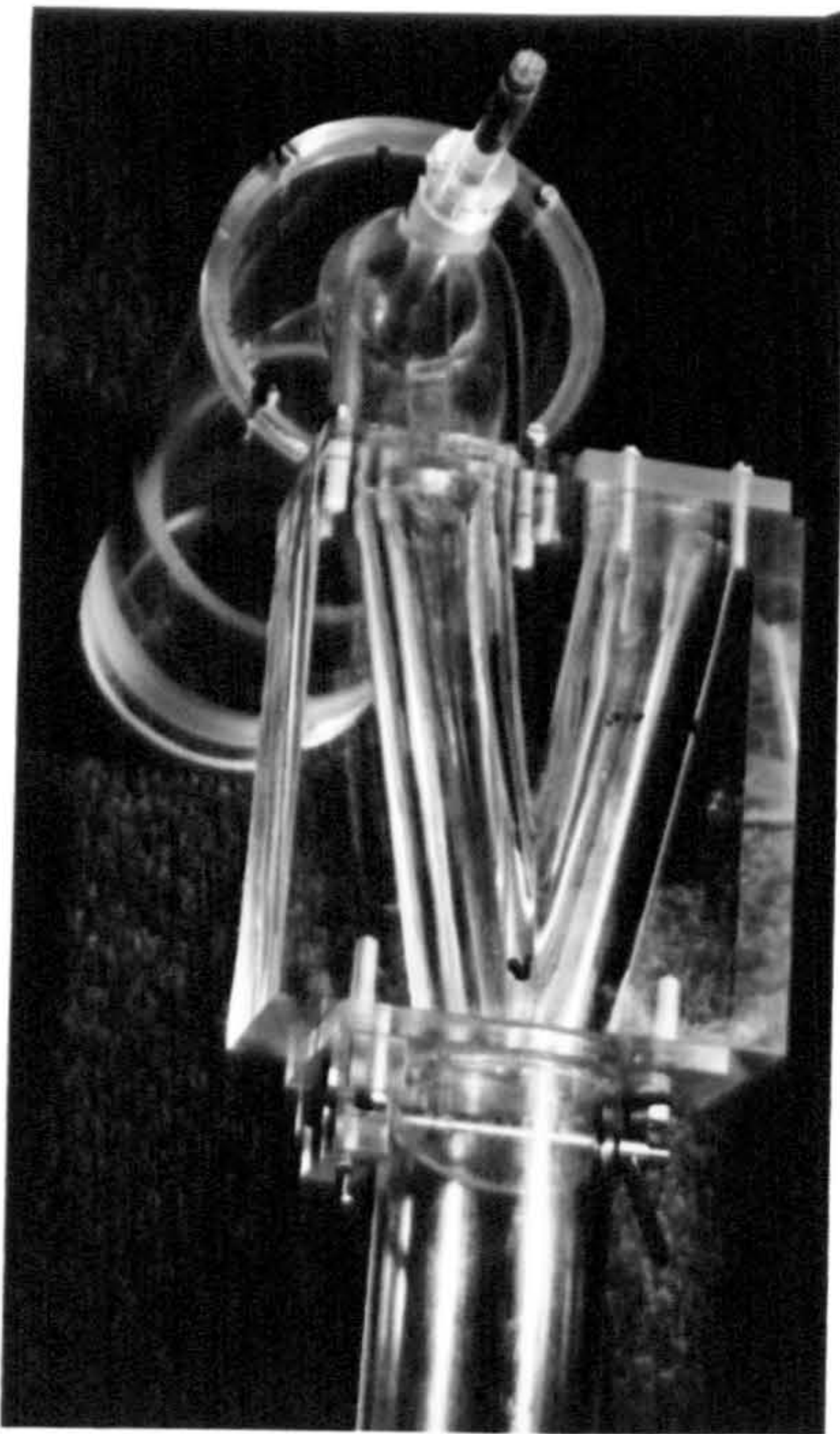


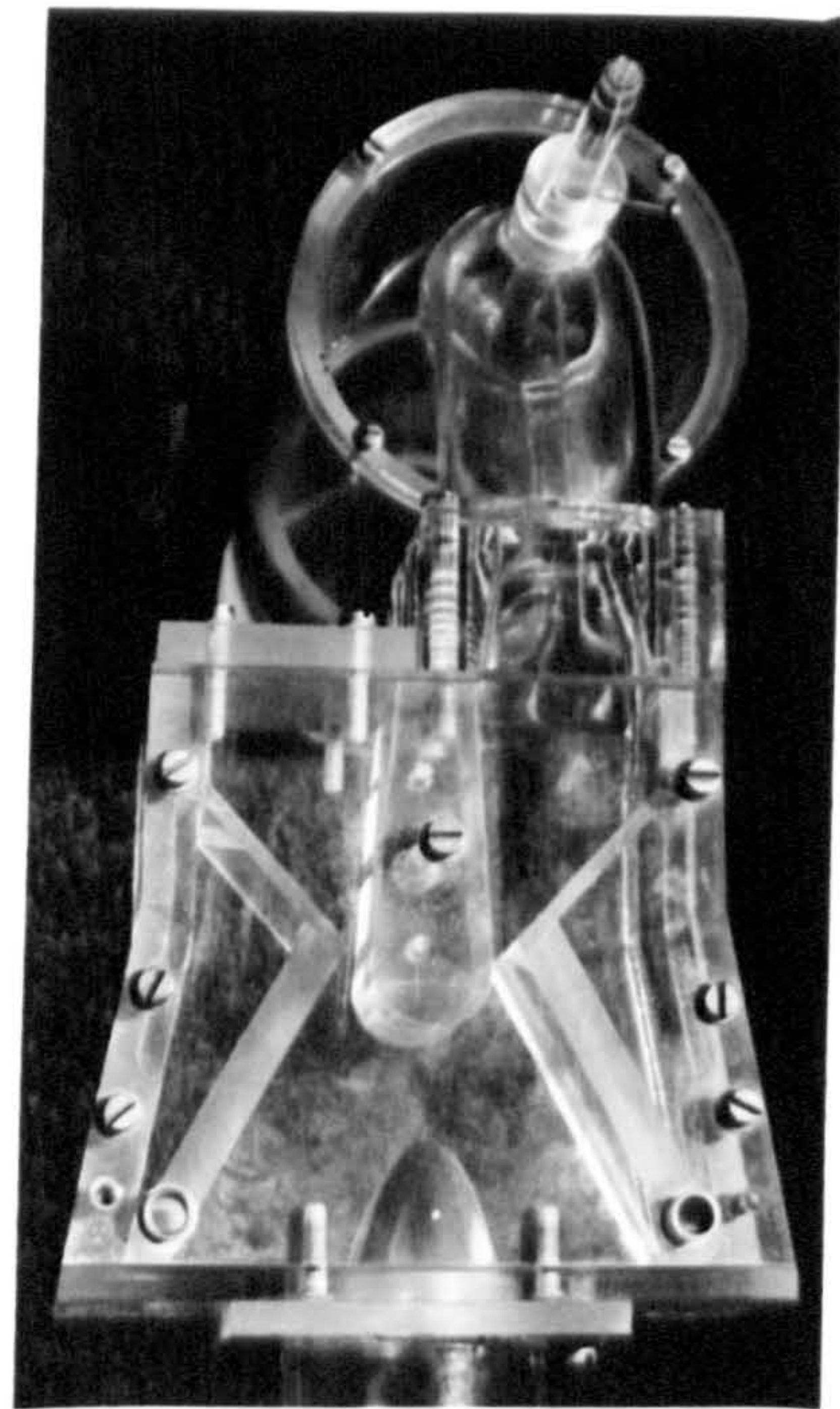
Fig. 4.3 The Perspex model of the cylinder and inlet valve.

is achieved by interchangeable Perspex ramps, which replace the spring loaded ramp system used in the actual device. These Perspex ramps were designed to cover a range of throttle settings. A typical ramp geometry representing a high* throttle setting can be seen in Fig. 4.2.b. These ramp settings and the corresponding butterfly valve angles will be discussed in detail at a later stage.

The arrangement of the assembled models of the device and the cylinder can be seen in Fig. 4.4.



(a) Butterfly system



(b) Variable ramp system

Fig. 4.4 Perspex models of the assembled systems.

In addition, the transparent models have to be encased in a Perspex box containing water in order to facilitate the observation of the flow in the models through curved or inclined walls. A typical arrangement of the model in the water filled Perspex box can be seen

* small throttle opening

in Fig. 4.5.

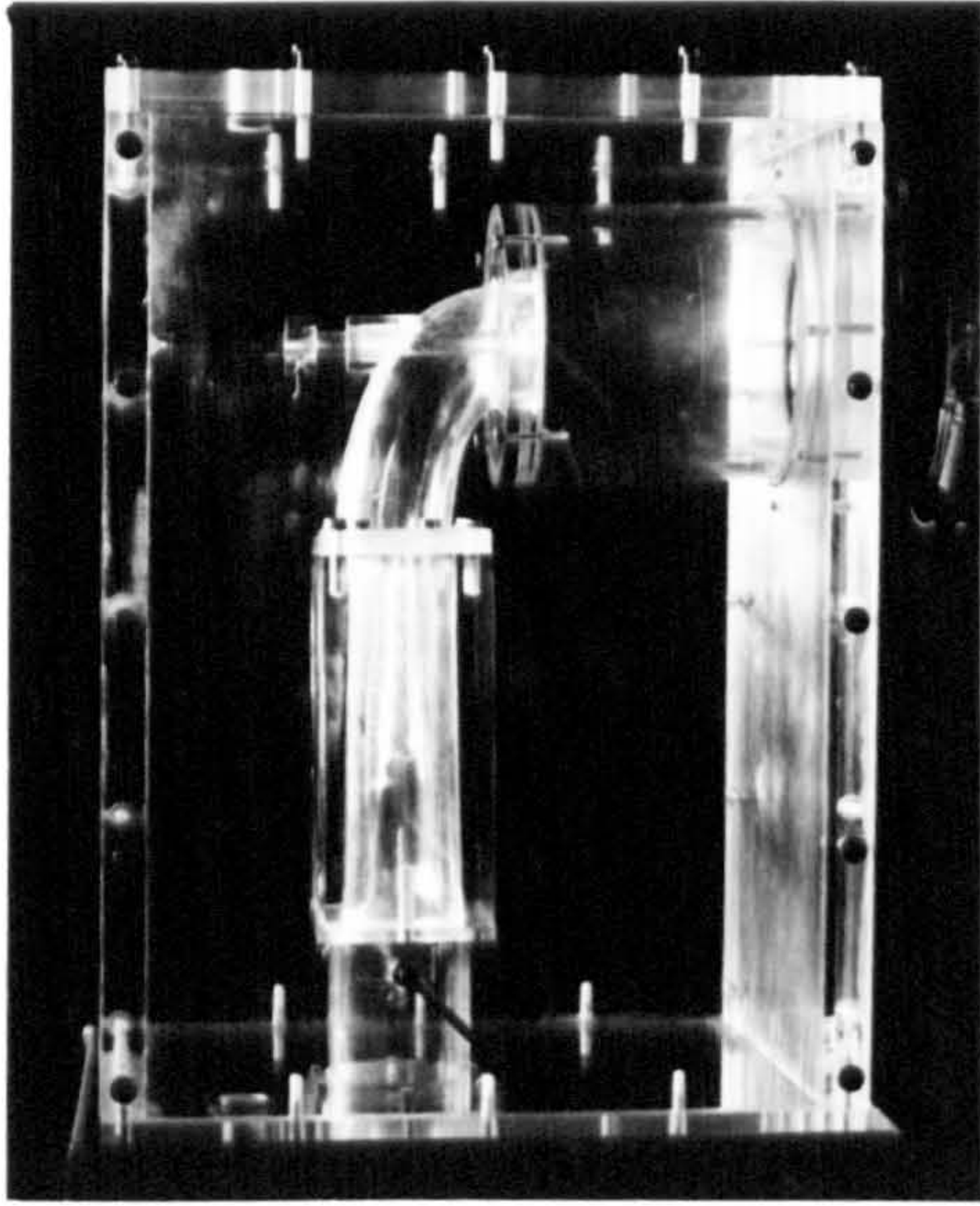
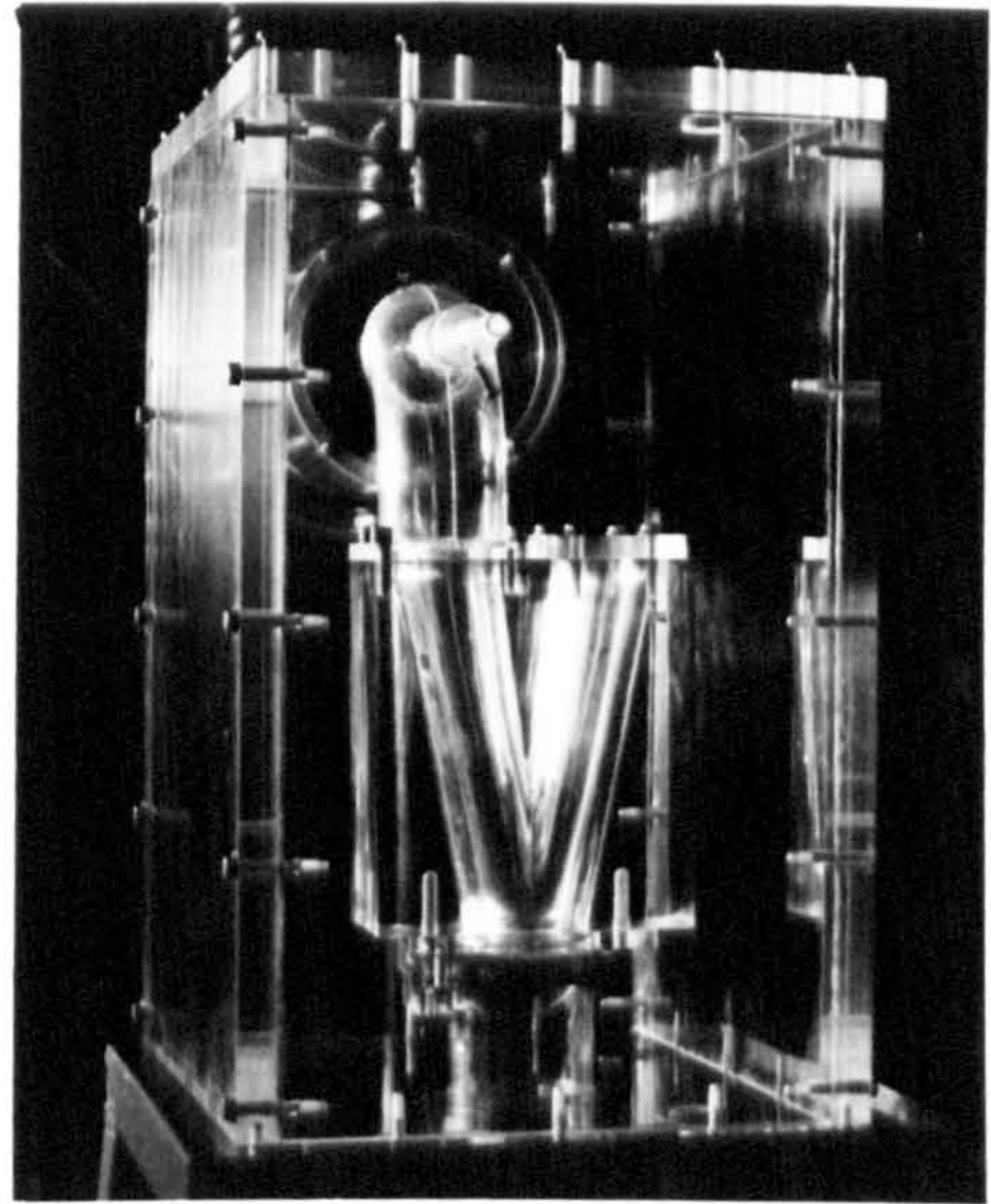


Fig. 4.5 Two views of a typical arrangement of model in water filled transparent box.

4.3.2 Lighting and photographic equipment for flat beam technique

In order to study the main features of flow field by direct observation or photography through the Perspex model, the air bubble tracers must be illuminated by a suitably powerful light source. Tracers of this kind, which can be observed by reflection, are particularly useful for the visualization of truly three-dimensional flows, because it is then possible to employ a selective lighting technique in which only a portion of the flow field is observed. In the flat beam technique, a narrow flat beam is projected through the transparent model, illuminating the tracers over a thin slice through the field. The flow can then be observed under two-dimensional conditions over

a plane and at any location within the complete field of flow, without obstruction by the intervening flow.

The flat beam technique requires a suitable type of lighting system. This usually takes the form of either a tubular light source or a bank of small lamps housed in container, with a system of slits to produce a narrow, clearly defined flat beam. The lighting system used in the present study was of this type. It was constructed at Cranfield and has been used successfully for similar studies over many years (Fig. 4.6). This particular equipment uses a powerful (3KW) tungsten lamp, which is continuously air-cooled by an axial fan installed in the housing.

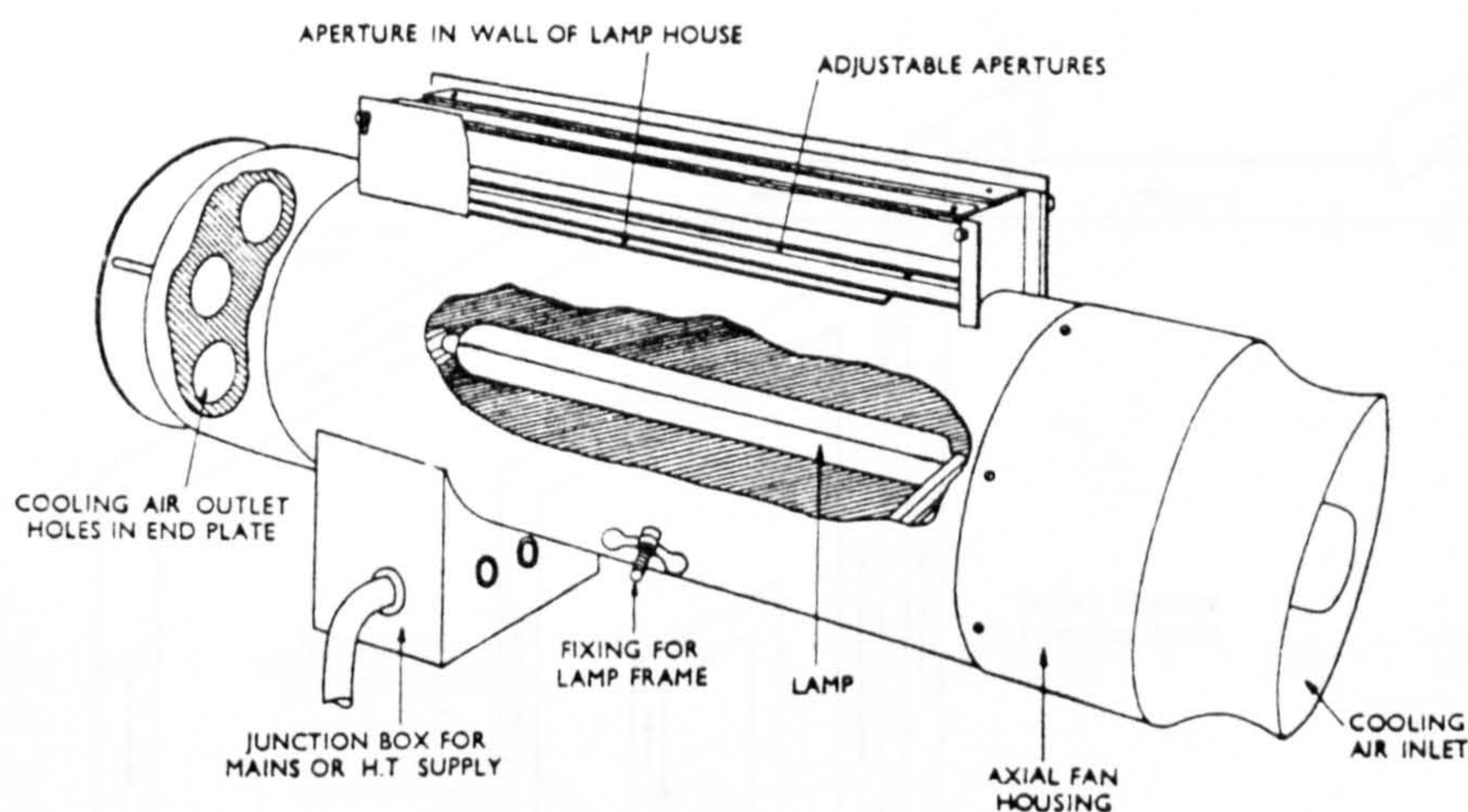


Fig. 4.6 Lighting equipment for flat beam technique

The width of the flat beam is controlled by the adjustable apertures (Fig. 4.6). This type of lighting system is ideal for use with water models. The continuous tungsten light source allows for comfortable observation of the flow in any chosen section of the model and from any viewing angle. Hence, a variety of views can be photographed routinely using this system.

The photographic equipment used consisted of a Nikon F1 SLR

camera body with a 50 mm 1.4 Nikkor lens. The camera was chosen for its range of exposure timings and its light metering, and the lens for its optical quality and range of apertures.

4.3.3 Arrangement of the water model network

The water network used for the flow visualization tests was of the closed-circuit nature, as illustrated in Fig. 4.7. Air bubbles are generated in the water tank through the impact between recirculating water jets and the free surface of water in the tank. Bubble-laden water is then pumped through the system, the flow of which is controlled by two metering and valve systems. One of them controls the total

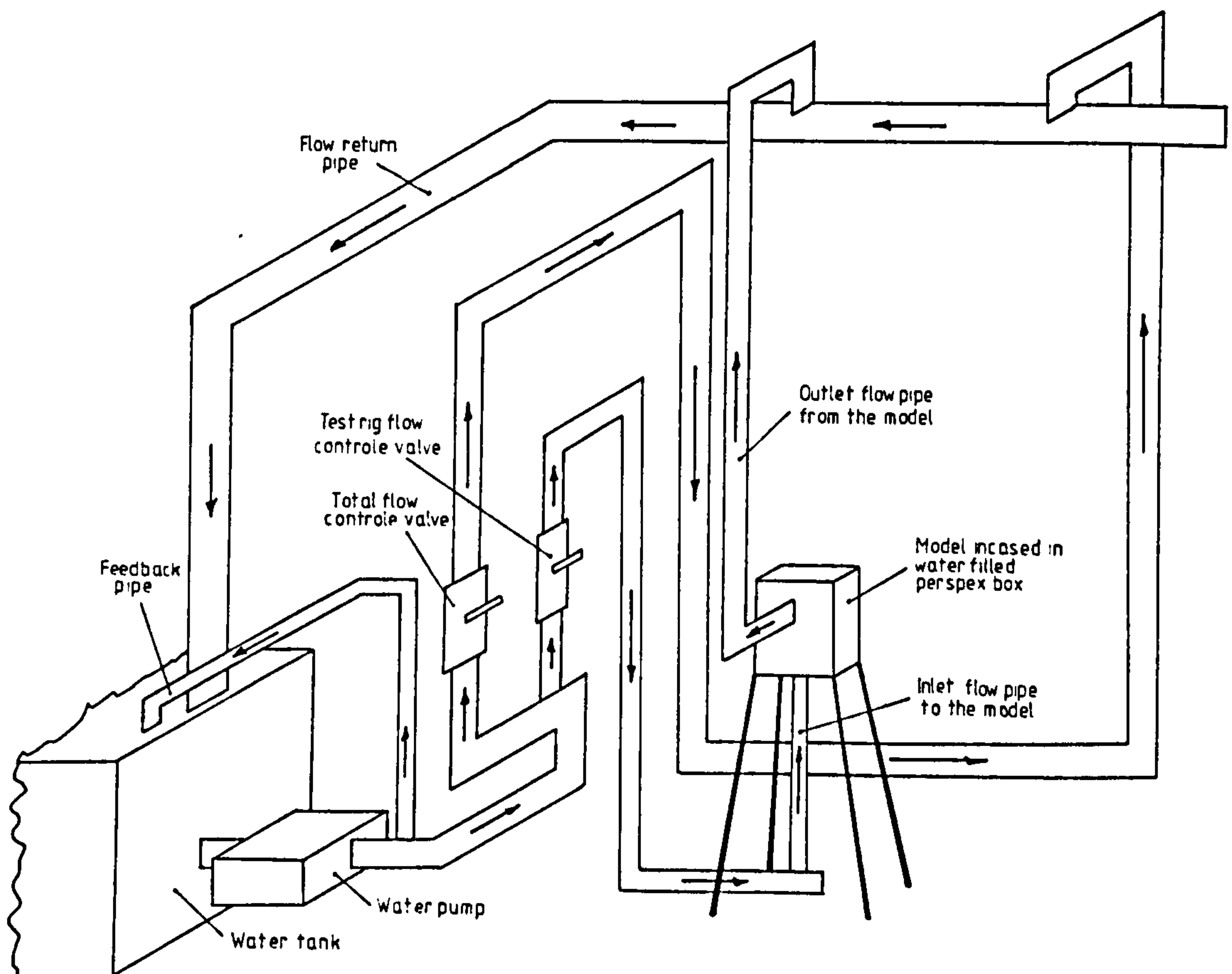
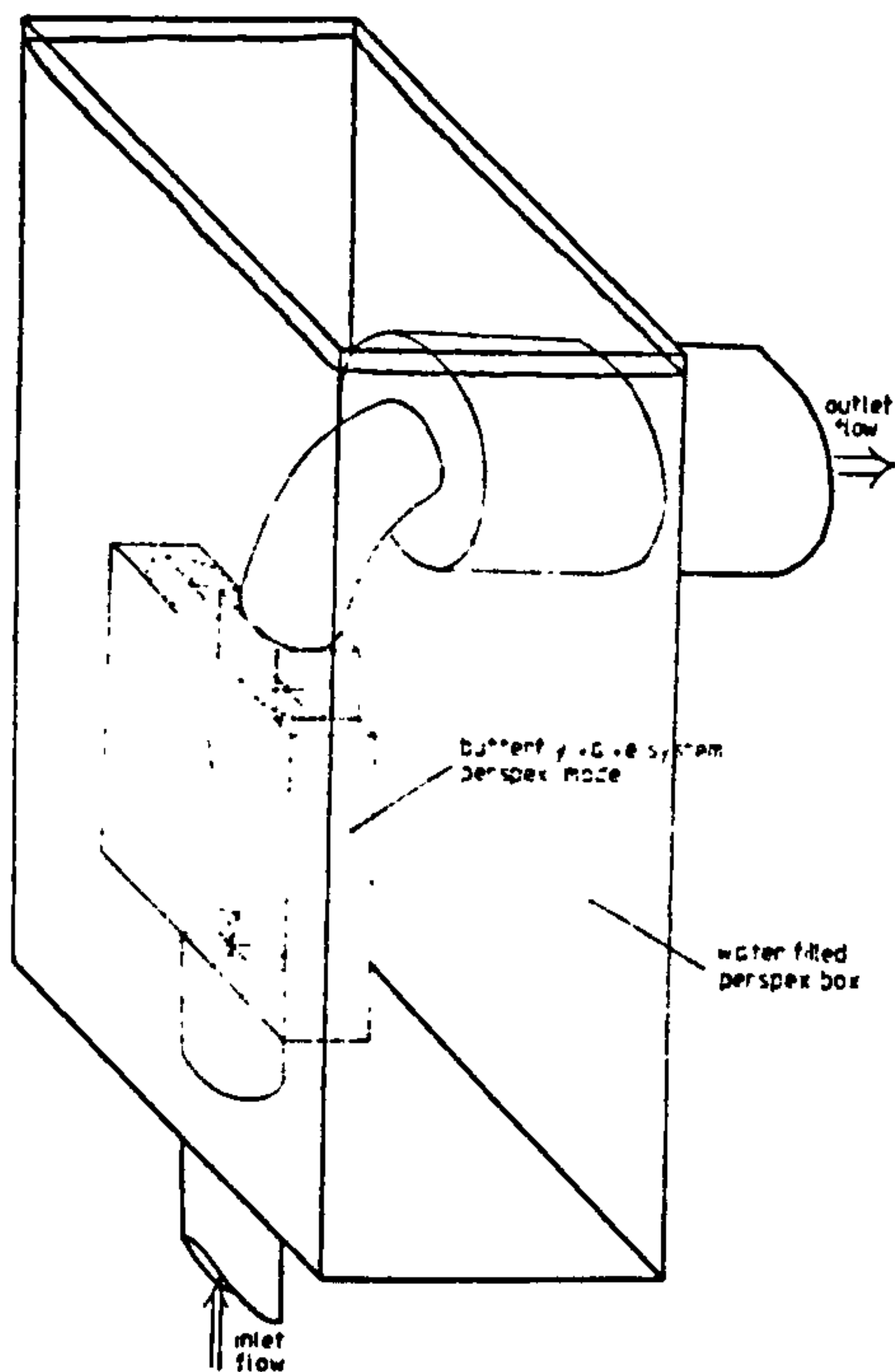
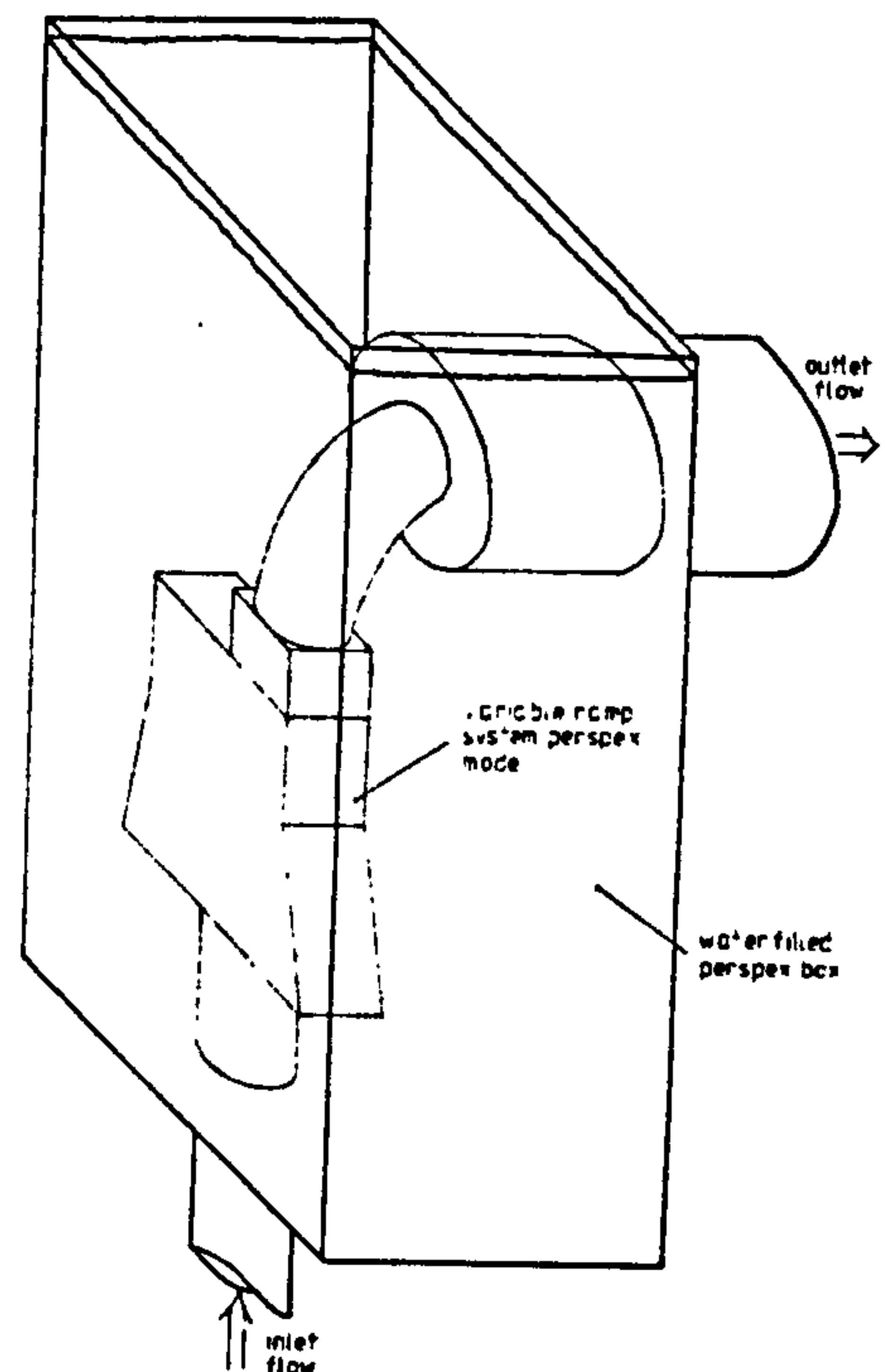


Fig. 4.7 Arrangement for water modelling network.

flow through the network and the other the rate of flow through the Perspex model (Fig. 4.7). Bubble-laden flow enters the Perspex model through the inlet pipe and leaves the model through the cylinder, as shown in Fig. 4.8.



(a) Variable ramp model



(b) Butterfly valve model

Fig. 4.8 Arrangement of Perspex models in the water network.

Water from the model, and surplus water through the total flow valve, is then deposited back into the reservoir tank via the flow return pipe. There is, therefore, a constant supply of bubble-laden water to the Perspex model.

4.4 Experimental procedure

4.4.1 Throttle settings

Comparisons of the flow field features between the models of the two systems were made on the basis of their effective throat area. In the variable ramp device, throttling is achieved through spring-loaded rotating flaps which change the nozzle throat area. In the Perspex models this latter mechanism is replaced by a set of fixed Perspex ramps (Fig. 4.9.a). Four different ramp geometries were considered for the flow visualization tests, and they covered

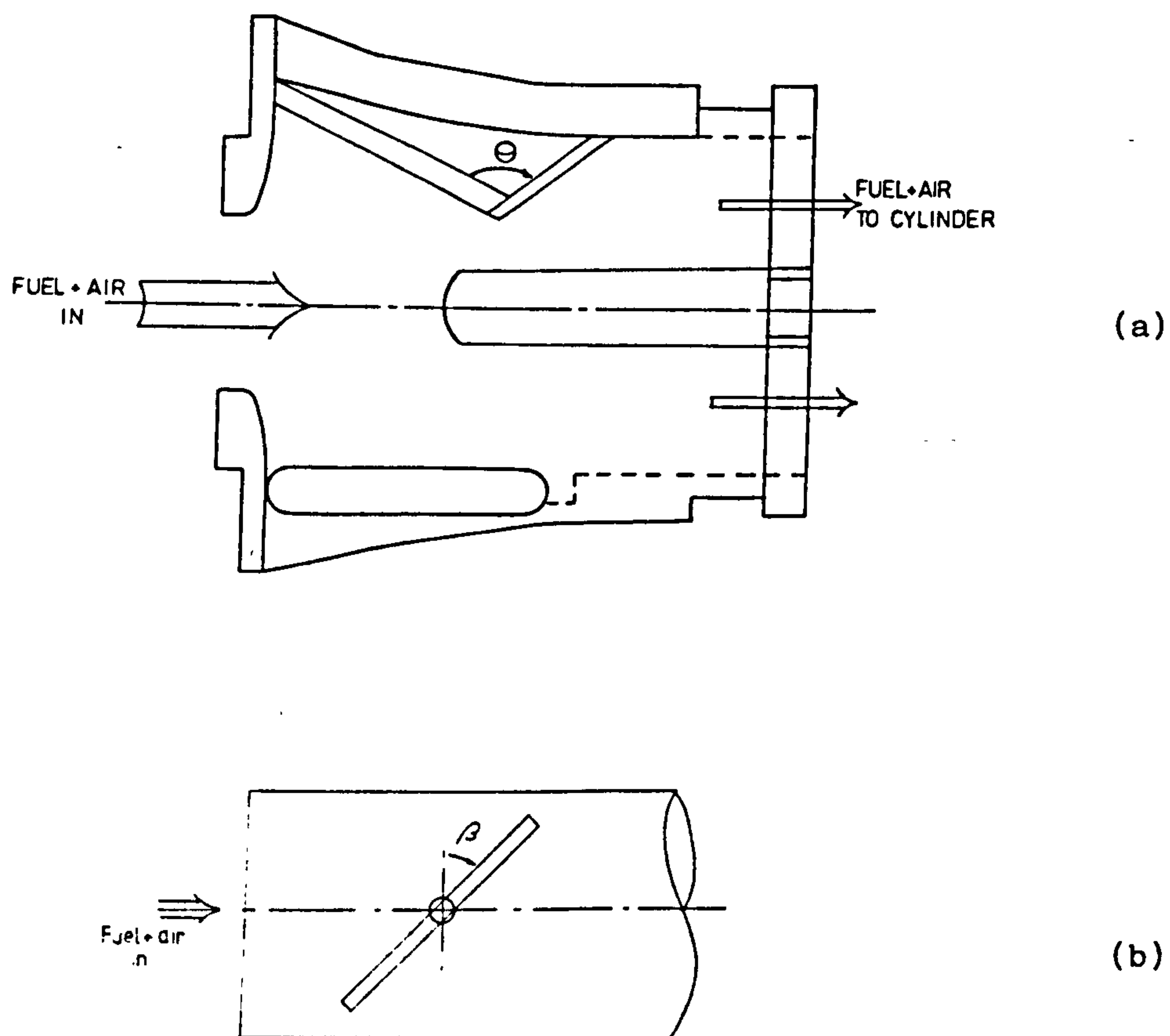


Fig. 4.9 Models of the throttling devices: (a) Variable ramp model, and (b) Butterfly valve model.

a range of throttling settings shown in Table 4.1. These settings will be referred to by included angle, Θ , the angle produced between arms of the ramp (Fig. 4.9.a).

In the butterfly valve models, effective throat areas corresponding to the variable ramp settings were produced by the rotation of the butterfly valve to appropriate angles (Fig. 4.9.b). These settings will be referred to by the angle, β , the angle created by the butterfly valve and the normal to the inlet flow (Table 4.1).

Test No.	Ae/Ai	Ramp Included Angle Θ	Butterfly Valve Angle β
1	0.13	110°	30°
2	0.36	130°	50°
3	0.58	150°	65°
4	WOT	180°	90°

Table 4.1 Throttle settings tested for the variable ramp and butterfly valve models, on the basis of effective throat area.
Ae = effective throat area ; Ai = inlet pipe flow area

4.4.2 Flow field visualization test procedures for the variable geometry ramp throttling model

Four ramp geometries were tested for the variable ramp model (Table 4.1). In each case, the appropriate fixed ramp was placed in the system. The model was then encased in the water filled Perspex box and placed in the water network (Fig. 4.8.b). Bubble-laden water was then pumped through the model. Appropriate planes were illuminated by the lighting system described earlier, and photographed.

The rates of water flow through the models were based on the average flow rates encountered in an engine. The engine considered was similar to that tested by Abbas (48) and Abdulla (49) in their studies of the throttling devices (1500 cc four-cylinder). The Reynolds numbers were based on average flow rates through the induction manifold at different engine speeds and the diameter of the inlet pipe. These were calculated to be of the order of 10^4 or more for all practical engine speeds. However, as mentioned before, when the Reynolds number is large enough throughout the system, then it is not necessary to insist on its exact identity. Over a wide range of flow conditions tested, the structure of flow through the models was found to be independent of flow rate or Reynolds number. Therefore, no attempt was made to represent average engine flow rates by detailed matching of Reynolds numbers for the water analogy tests. This allowed a certain measure of freedom in the choice of flow rates through the models for visualization purposes.

Clarity and quality of the pictures presented of the flow field by the air bubble tracers depends to a large extent on their number and size. To a large degree this is controlled by the rate of flow and the surface tension of water in the storage tank (Fig. 4.7). The size of the air bubbles in particular is directly related to the surface tension of the water in the water tank. In the tests this was controlled by the introduction of a very small amount of washing liquid into the water tank, in order to reduce water surface tension and hence air bubble size. The distribution and density of the bubbles was, however, controlled by two metering and flow control systems (Fig. 4.7). One valve controlled the total flow rate through the network, and hence the total number of bubbles. The ideal proportion of the bubble-laden water was then diverted through the models by the other metering and valve system.

For all model configurations 40 gallons per minute was found to be the ideal total flow rate through the network, in terms of producing the optimum number of bubbles. 10 gallons per minute was found to be the ideal rate of flow through the actual models. This flow rate not only satisfied the Reynolds number requirement (i.e. $Re \gg 10^4$), but in addition provided the best picture of flow field fea -

tures for photographic and observation purposes.

For each ramp geometry configuration tested, four planes were considered for the purpose of photography (i.e. planes A, B, C and D) (Fig. 4.10). These planes were illuminated in turn by a 3 mm thick,

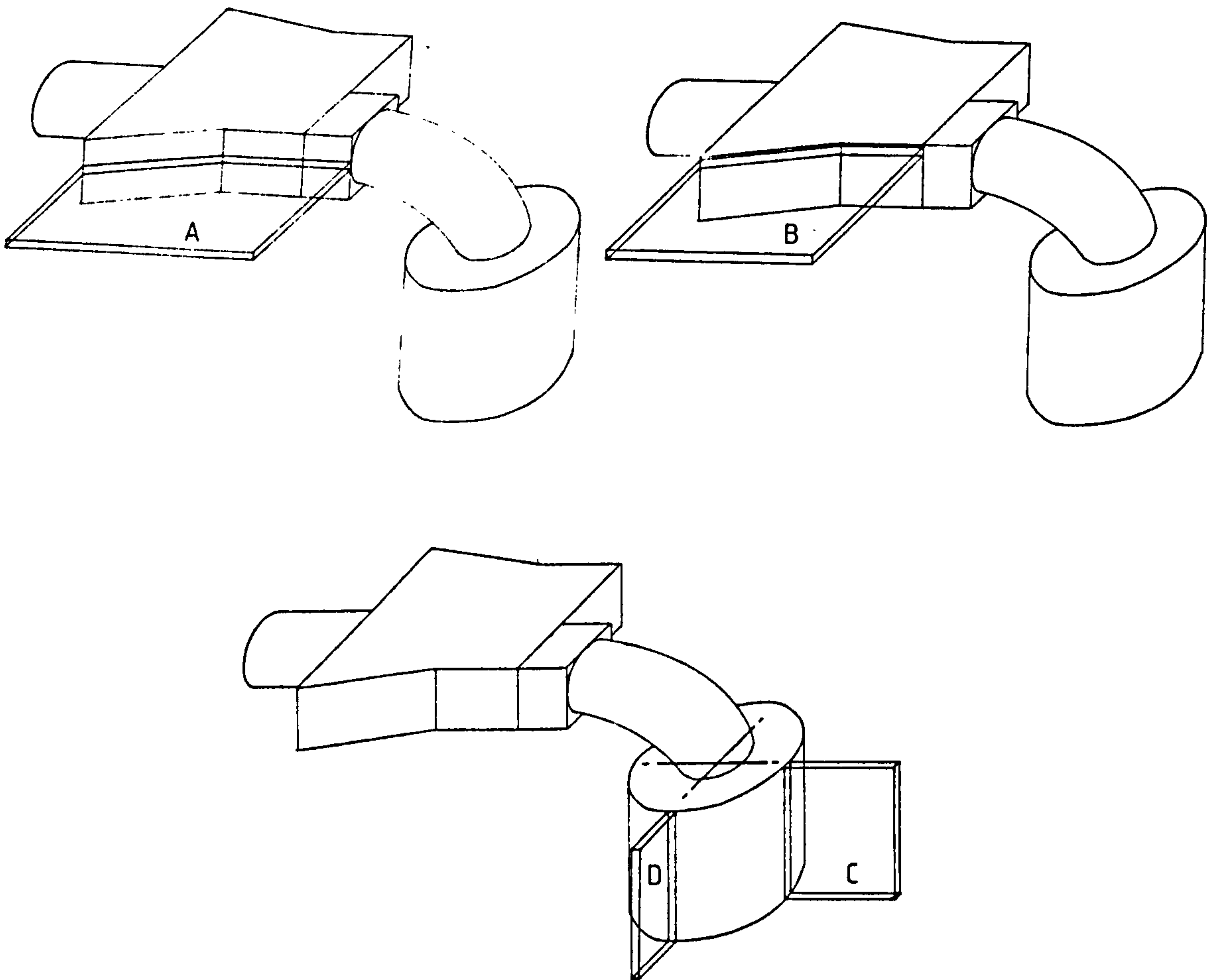


Fig. 4.10 Photographed planes of flow in the variable geometry ramp model.

high intensity light sheet produced by the flat beam lighting system outlined previously (Fig. 4.6). This particular thickness of light sheet was found to be satisfactory both in terms of providing a clear picture of the flow field and representing, by the depth of illumination, a two-dimensional picture of the flow in a three-dimensional flow field.

Photographs of the planes were taken in turn using an SLR camera and viewing from a right angle to the planes of illumination. Each plane was photographed for a number of exposure times and aperture settings, with the aid of the built-in light metering system. These exposure times ranged between 1/4 sec. to 1/30 sec. Photographs taken at 1/8 sec. were found to be best in terms of clarity of the tracers and the quality of the pictures. Photographs of the planes A, B, C and D for different ramp geometries can be seen in Plates 4.1 to 4.8.

These planes were mainly chosen because they provided a good view of the main features of the flow field and the general structure of mixing and turbulence processes taking place through the models. Planes A and B were of particular interest for they provide valuable information about the effects of ramp angle, nozzle throat area and wall friction on mixing and recirculation, whereas the effect of ramp angle on cylinder flow can be studied through planes C and D.

4.4.3 Flow visualization tests for the butterfly valve throttling model

The experimental procedure for the butterfly models was identical to that of the variable ramp models. Four different throttle settings were tested by rotation and locking in place of the butterfly valve to appropriate angles. These angles were based on setting the effective throat areas equal to that of variable ramp systems tested, and can be seen in Table 4.1.

For each butterfly valve setting five planes were photographed (i.e. A', B', C', D' and E') (Fig. 4.11). Planes A' and B' were chosen because they provide valuable information about the effect of butterfly valve angle on recirculation, mixing and distribution of flow. Plane C' provided an indication to the extent by which the mixing and turbulence persist downstream of the butterfly valve. In addition, the effect of various values of angle on in-cylinder flow was studied through planes D' and E'. Photographs of these planes can be seen in Plates 4.9 to 4.16.

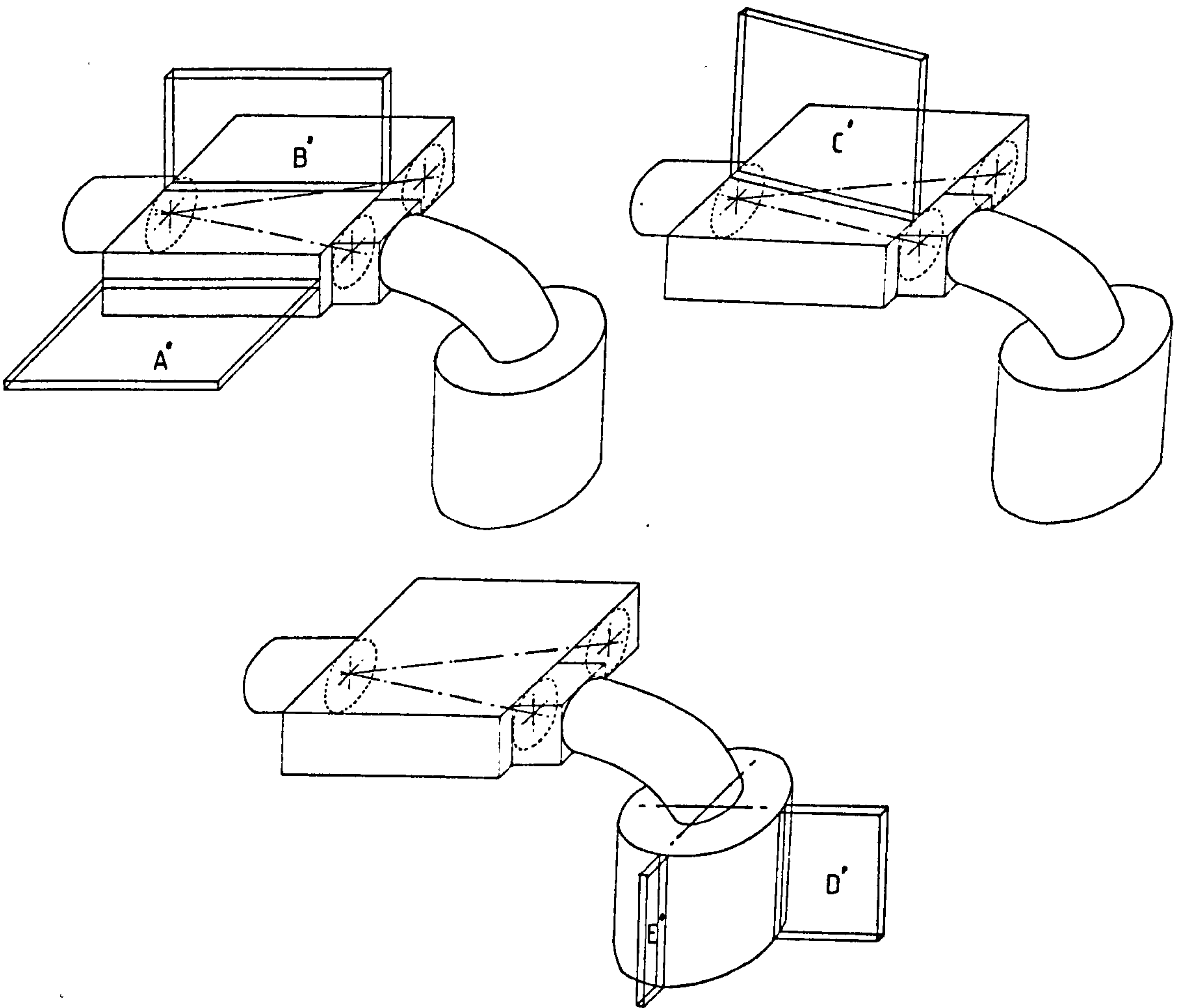


Fig. 4.11 Photographed planes in the butterfly valve throttling model.

4.5 Discussion of results

4.5.1 Variable geometry ramp results

4.5.1.1 Ramp angle $\Theta = 110^\circ$

Photographs of the flow pattern for the variable geometry model at ramp angle $\Theta = 110^\circ$ can be seen in Plates 4.1 and 4.2. Photographs A and B represent flow fields through the mid-plane and the plane near the wall of the throttling section, respectively. As can be seen from the photographs, there are three major mixing and recirculation zones. Two of these recirculation zones appear immediately

after the flow enters the throttling section, as the result of the sudden expansion. In this particular configuration these recirculations appeared to have little effect on mixing downstream of the ramp. On the other hand, the steep ramp angle at this low throttle setting produces a strongly convergent divergent passage, resulting in a particularly pronounced recirculation zone downstream of the ramp. The nature of flow in the throttling section to a large extent was two-dimensional. However, as can be seen in Plate 4.1 there are significant differences in flow structure between planes A and B. These differences were mainly as a result of inlet and outlet geometries of the section. Flow enters the throttling section through a circular plane and leaves through a rectangular to circular cross-sectional adaptor. In plane B the circular nature of the inlet plane results in the reduction of inlet flow width through this plane in comparison with plane A, resulting in a smaller recirculation zones immediately after the flow enters the section for plane B in comparison with plane A. The reduction in size of the recirculation zone downstream of the ramp in plane B was mainly due to the gradual reduction in the flow width through this plane as a result of the adaptor.

The extent of downstream mixing and its effect on cylinder flow can be seen in Plate 4.2. As shown in plane C, mixing persists downstream of the ramp right up to the cylinder inlet port. On the other hand, it is difficult from plane D to identify the extent to which upstream mixing affects the in-cylinder mixing process. This was mainly due to vortex instability in the cylinder section, as the result of piston absence in the cylinder to stabilize and hold the vortex structure created by the flow around the inlet valve. However, as can be seen in plane D, flow enters the cylinder at different angles through the inlet valve. This indicates pronounced velocity variations around the inlet valve, which to a large extent is due to large velocity gradients created by the ramp upstream of the inlet valve. Visual observations indicated higher rates of in-cylinder mixing and turbulence as a result of upstream mixing and velocity variation around the inlet valve.

4.5.1.2 Ramp angle $\Theta = 130^\circ$

Plates 4.3 and 4.4 contain photographs of the flow fields for the variable ramp model at ramp angle $\Theta = 130^\circ$.

As is shown in Plate 4.3 flow fields through planes A and B still contain three major recirculation zones. However, the recirculation zones created as a result of flow entering the throttling section appear to be larger and stronger in comparison with the previous test case ($\Theta = 110^\circ$). To a large extent this is due to the increased flow cross-sectional area, as a result of shallower ramp angle. On the other hand, the recirculation zone downstream of ramp appears to be smaller and weaker in comparison with the test case $\Theta = 110^\circ$. This is largely due to the less pronounced convergent-divergent passage, larger effective throat area and less severe adverse velocity gradient downstream of the ramp.

It can be seen from the flow field through plane C (Plate 4.4) that mixing and turbulence persist downstream and in particular up to the cylinder inlet port, but to a lesser extent in comparison to the previous case ($\Theta = 110^\circ$) tested. This is apparent from the clearer way in which stream lines are distinguishable near the cylinder in the $\Theta = 130^\circ$ case. In addition flow appears to be more evenly distributed around the inlet valve, as it enters the cylinder. This is apparent from the flow entry angles in plane D, which are more evenly matched.

4.5.1.3 Ramp angle $\Theta = 150^\circ$

Photographs of the flow field at ramp angle $\Theta = 150^\circ$ can be seen in Plates 4.5 and 4.6. Photographs of planes A and B indicate that there is little change in the nature of recirculation zone, immediately after the inlet plane, on the induction side of the throttling section (i.e. right corner of the throttling section). On the other hand, the structure of the recirculation zone is somehow different on the stagnant side of the throttling section, and tends to persist into the stagnant induction manifold. The mixing created by the small recirculation zone downstream of the ramp is much weaker in strength than the previous cases. There is very little mixing

further downstream near the cylinder. This is shown in the photograph of the plane C by the clarity the stream lines can be distinguished. The flow around the inlet valve also appear to be fairly evenly distributed.

4.5.1.4 Ramp angle $\Theta = 180^\circ$ (wide open throttle)

Views of the flow field for the variable ramp model at wide open throttle can be seen in Plates 4.7 and 4.8. Planes A and B show that the structure of the flow field in the throttling section is very similar to the previous test case ($\Theta = 150^\circ$), with the exception that the recirculation or mixing zone downstream of the ramp is absent. Planes C and D also show very little sign of mixing further downstream, near the cylinder.

4.5.2 Butterfly valve results

4.5.2.1 Butterfly valve angle $\beta = 30^\circ$

Plates 4.9 and 4.10 contain photographs of the flow field structure for the butterfly valve model at valve angle $\beta = 30^\circ$. This valve angle corresponds to a ramp angle $\Theta = 110^\circ$ for the variable ramp model in terms of effective throat area.

The effect of valve geometry and angle on flow structure, mixing and generation of turbulence can be seen in photographs of planes A' and B'. As can be seen, in plane B' the angle of incidence between the inlet flow and the butterfly valve results in the division of stream lines along the surface of the valve. As a consequence of this, about one third of the flow appear to pass through the passage created between the upstream edge of the valve and the inlet pipe, and the rest through the downstream passage. As a result of the high velocities through the valve throat area, a heavy recirculation and mixing zone is created downstream of the butterfly valve. However, mixing and turbulence appear to decay further downstream near the cylinder port. This, to a certain extent, can be seen in planes C' and D'. In addition, from plane E' it appears that flow enters the cylinder at fairly even angles around the inlet valve. In terms of

flow distribution, as can be seen in A', and as might be expected, there appears to be a pronounced tendency for the flow to recirculate strongly in the closed, and hence stagnant, induction pipe.

4.5.2.2 Butterfly valve angle $\beta = 50^\circ$

Photographs of the flow fields for the butterfly model at valve angle $\beta = 50^\circ$ can be seen in Plates 4.11 and 4.12. This valve angle corresponds to a ramp angle $\Theta = 130^\circ$.

As can be seen from planes A' and B', the recirculation and mixing zone downstream of the valve appears to be smaller and weaker than in the previous case tested ($\beta = 30^\circ$). In addition, turbulence and intense mixing tend to decay very rapidly further downstream. This can be inferred from the clarity of individual tracers, which are distinguishable in planes C' and D'. On the other hand, the recirculation in the stagnant induction pipe is also still very apparent in plane A'.

4.5.2.3 Butterfly valve angle $\beta = 65^\circ$

Plates 4.13 and 4.14 contain photographs of the flow fields for the butterfly model at valve angle $\beta = 65^\circ$. This angle corresponds to a ramp angle $\Theta = 150^\circ$.

From planes B' and A' there appears to be very little recirculation or downstream mixing created as a result of the butterfly angle. On the other hand, unlike previous cases, flow downstream of the junction between the induction pipes appears to be significantly affected by the recirculation in the stagnant induction pipe. This is evident in plane A' from the flow separation and recirculation zone downstream of the junction of the pipes. However, this upstream mixing tends to decay very rapidly. and as can be seen from planes C' and D' no trace of it can be observed downstream near the cylinder.

4.5.2.4 Butterfly valve at wide open throttle

Flow field photographs for the butterfly model at wide open throttle can be seen in Plates 4.15 and 4.16.

As can be seen from planes A' and B', there is virtually

no mixing or recirculation created as a result of the valve geometry. The only sign of mixing appears downstream of the junction and it appears to be the result of flow recirculation in the stagnant induction pipe. However, this appears to have little or no effect on the flow downstream near the cylinder, as can be seen in planes C' and D'.

4.5.3 Comparison of the results of the two alternative systems

The flow visualization tests appear to show that the steep ramp angles at high throttle setting produce significantly stronger recirculation zones and intenser downstream mixing, in comparison with the butterfly valve, on the basis of comparable effective throat area. However, these advantages tend to diminish at lower throttle settings and, to a large extent, disappear at wide open throttle.

In the variable ramp model, at high throttle settings, the high velocity gradients created immediately downstream of the ramp, appear to persist further downstream, resulting in pronounced flow variations around the inlet valve. These velocity variations around the inlet valve appeared to encourage further incylinder mixing and turbulence. Such pronounced velocity variations were not encountered in the butterfly model at any throttle settings.

Throughout the throttling range, the variable ramp system showed little sign of flow diversion or recirculation in the stagnant induction pipe. However, in the case of the butterfly valve system, flow appeared to recirculate freely through the blocked manifold, over the complete range of throttle settings, and specially at low throttle settings, where the geometry of the manifold system rather than the butterfly valve determines the flow variation through the model.

4.6 Concluding remarks

The results obtained in this chapter can be extended only qualitatively to the actual engine. The flow structure in the models tested differs significantly from the engine flow. The absence of transient effects and a moving piston in the models can play an impor-

tant role in the flow field structure. This is especially true of the incylinder flow, where the absence of a piston causes vortex breakup and instability in the cylinder. However, the following conclusions can be drawn with a measure of confidence as applied to the models in question.

- General features of the flow for both systems were found to be substantially independent of flow rate, eliminating the need for exact identity of Reynolds number.

- In the case of the turbulence generating ramp, the steep ramp angles at high throttle setting, which produce a strongly convergent-divergent passage, results in a particularly pronounced recirculation zone with intense downstream mixing near the cylinder.

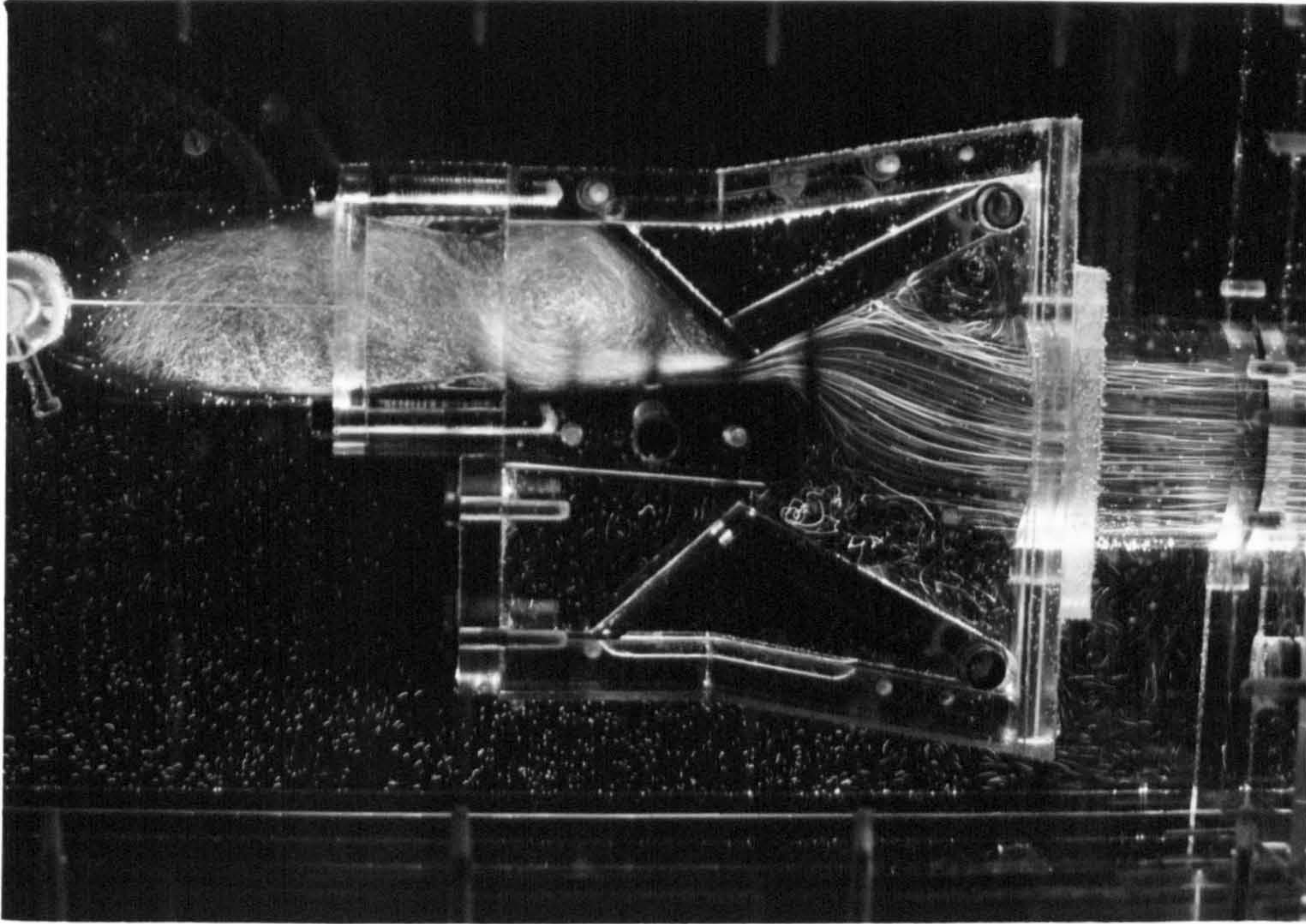
- The extent and persistence of the recirculation zone in the turbulence generating ramp tends to be diminished at lower throttle settings, and is completely lost at wide open throttle.

- Comparisons with the butterfly valve on the basis of effective throat area suggest that recirculation and mixing are significantly weaker in this more conventional device.

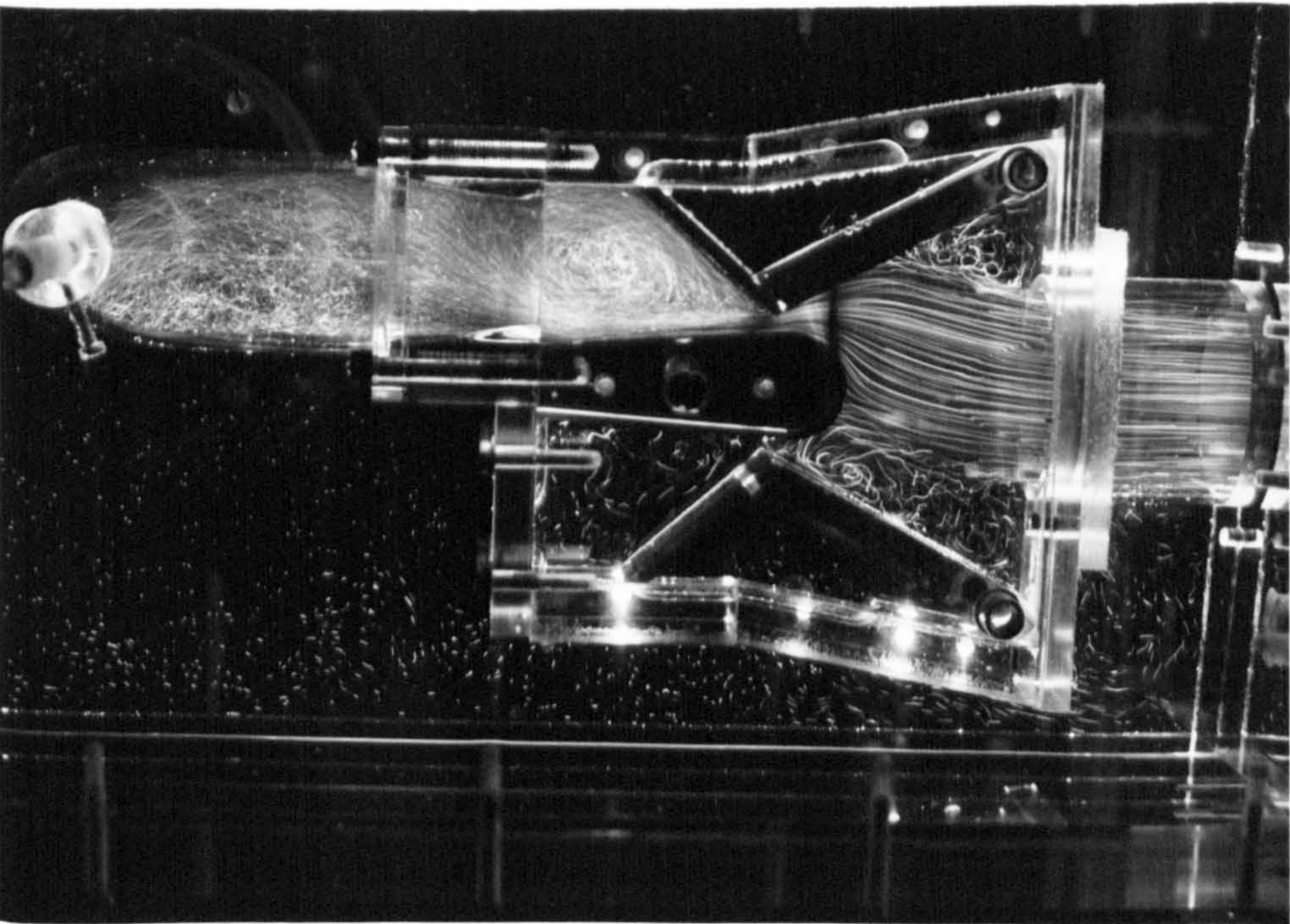
- The variable ramp device appeared to show better flow distribution characteristics in comparison with the butterfly valve system, by encouraging very little recirculation or diversion of the flow in the secondary induction manifold.

- In the context of engine performance, such observations could mean substantially better mixing and distribution of fuel and air in the induction manifold of an engine using the variable ramp device, and subsequently higher incylinder turbulence, resulting in higher rates of combustion and better overall engine performance.

In the following chapter the essentially qualitative data obtained from the water analogy tests are complemented by some hot wire velocity measurements at particular stations in the Perspex models, with air replacing the water as the flow medium.

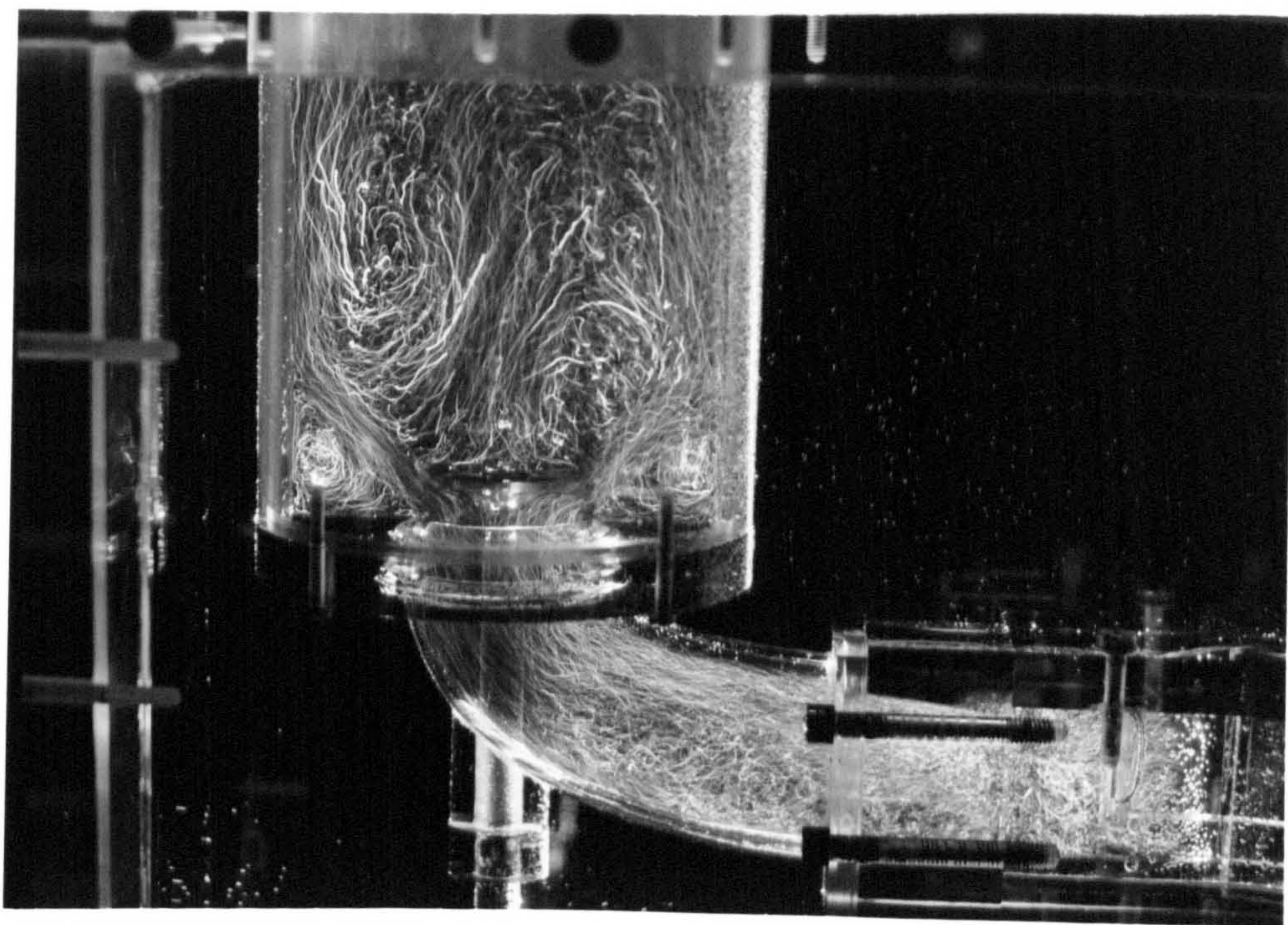


Plane B

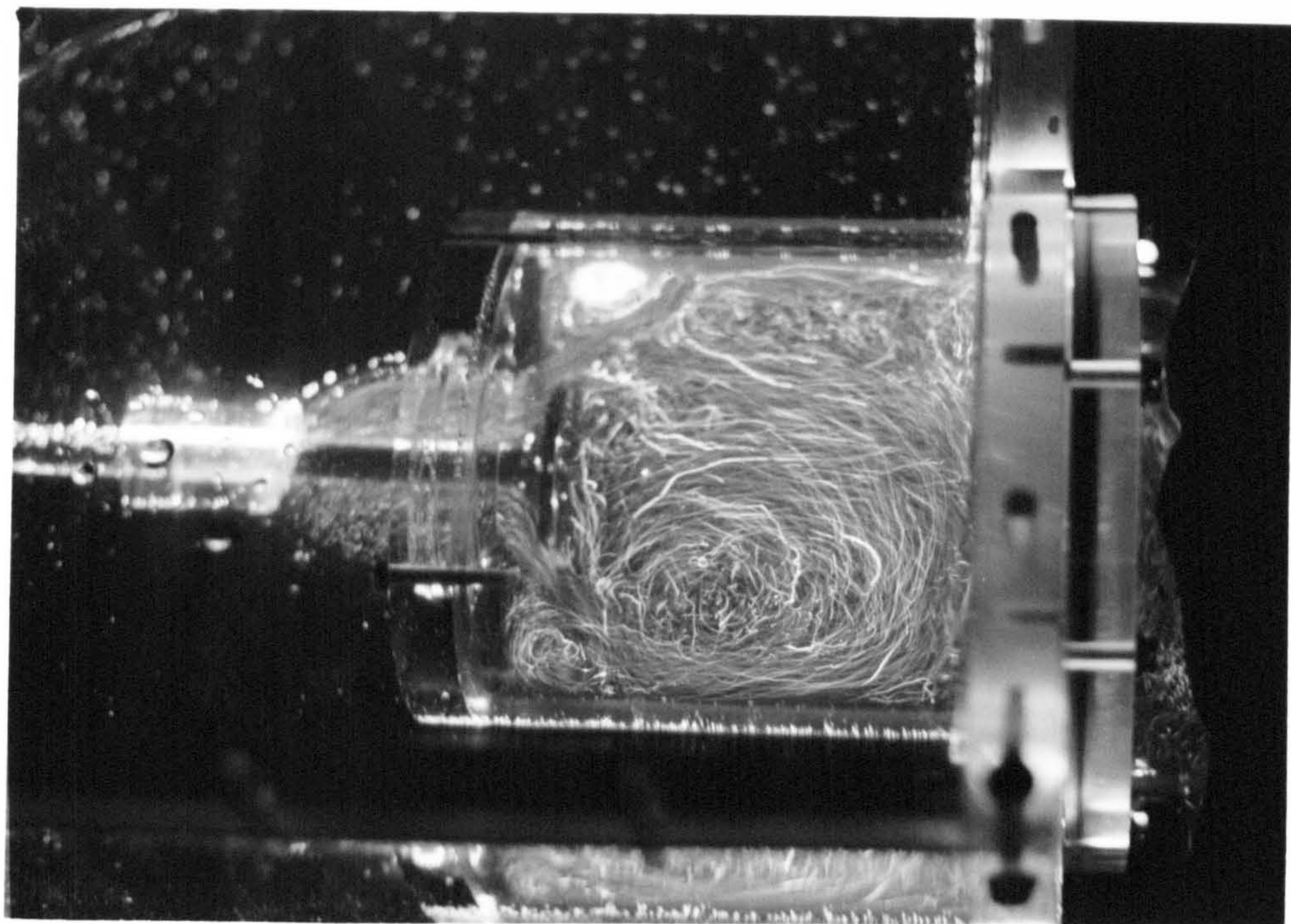


Plane A

Plate 4.1 Photographs of Planes A and B for variable geometry ramp throttling model at $\theta = 110^\circ$

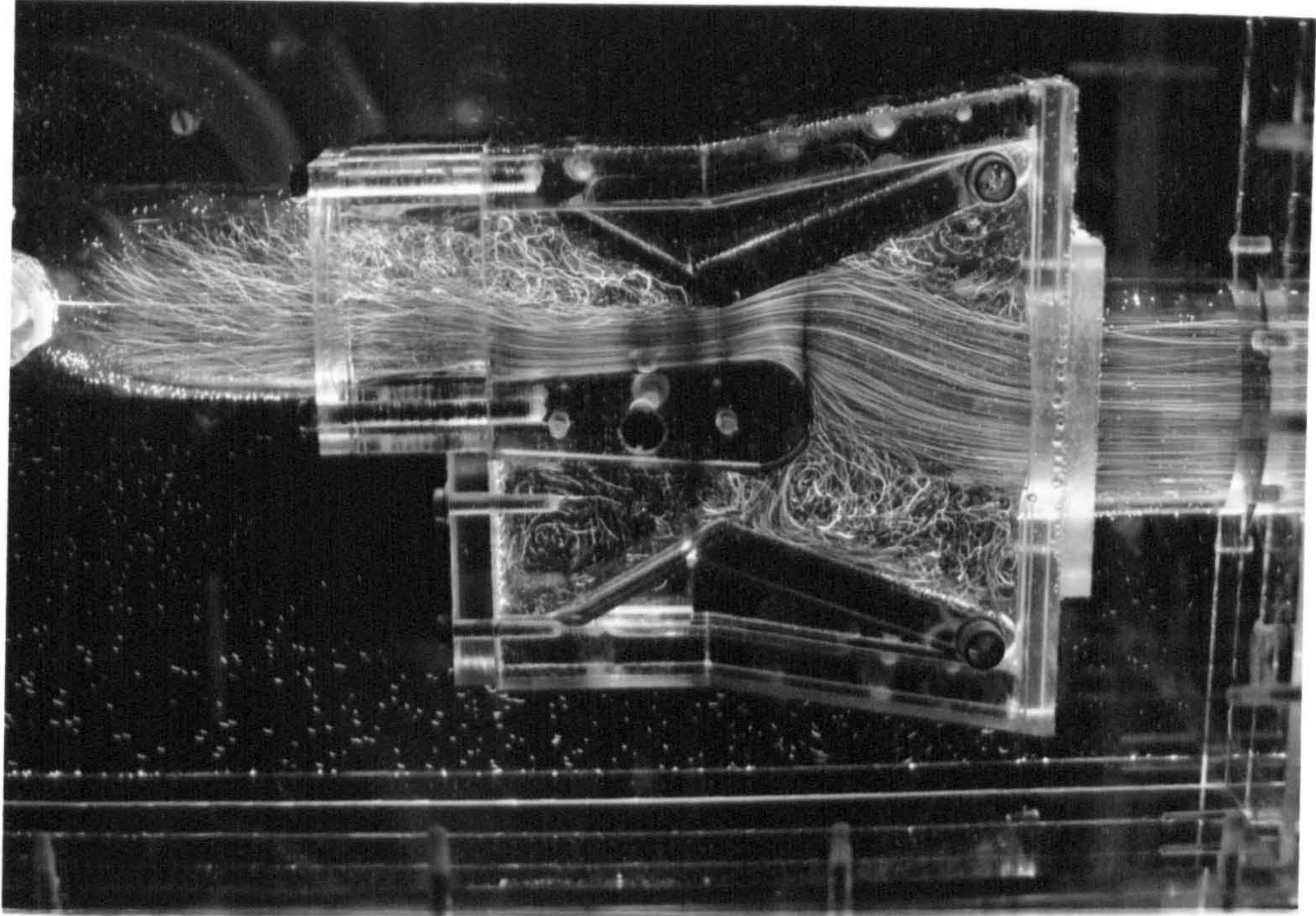


Plane C

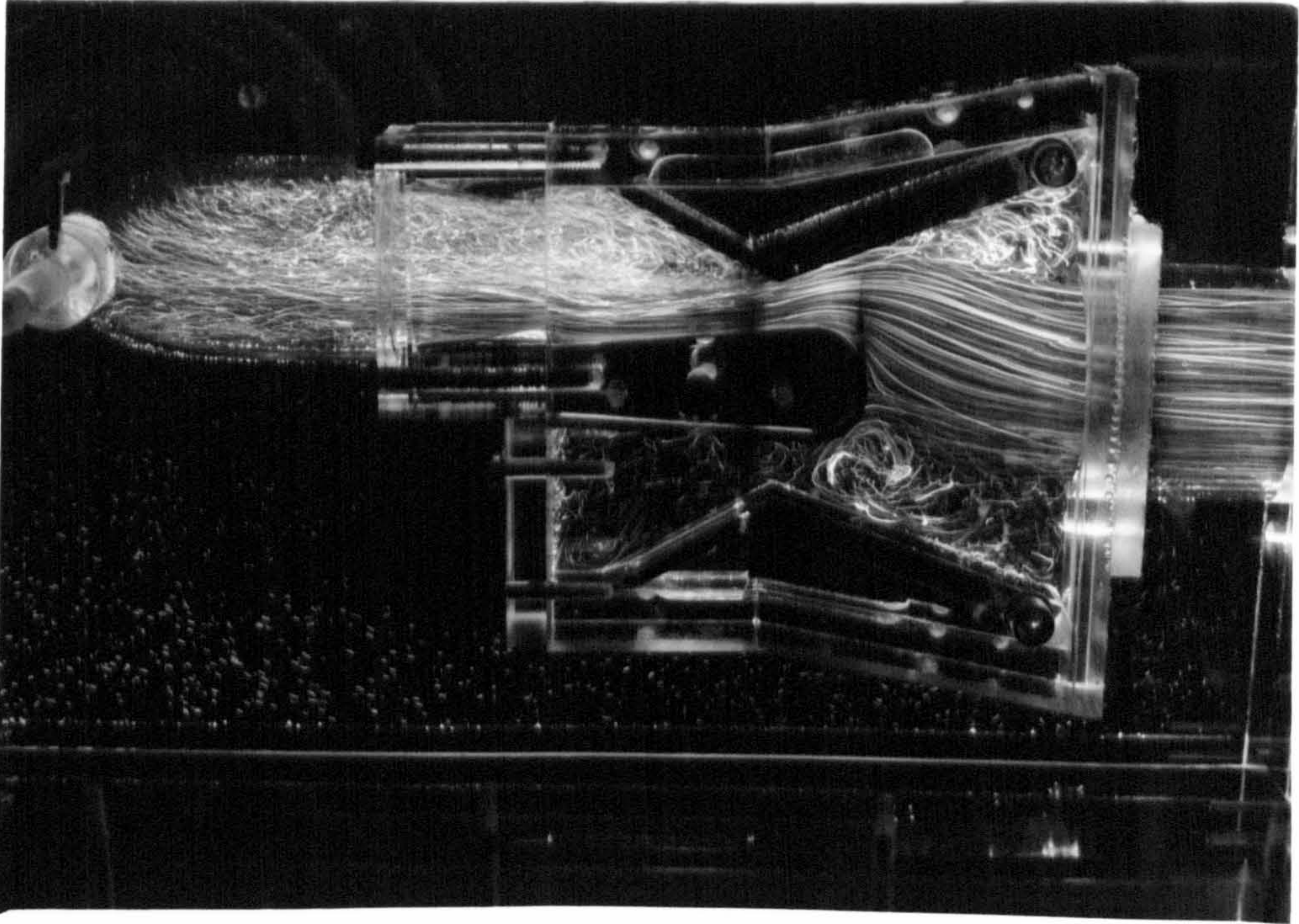


Plane D

Plate 4.2 Photographs of Planes C and D for variable geometry ramp model at $\Theta = 110^\circ$

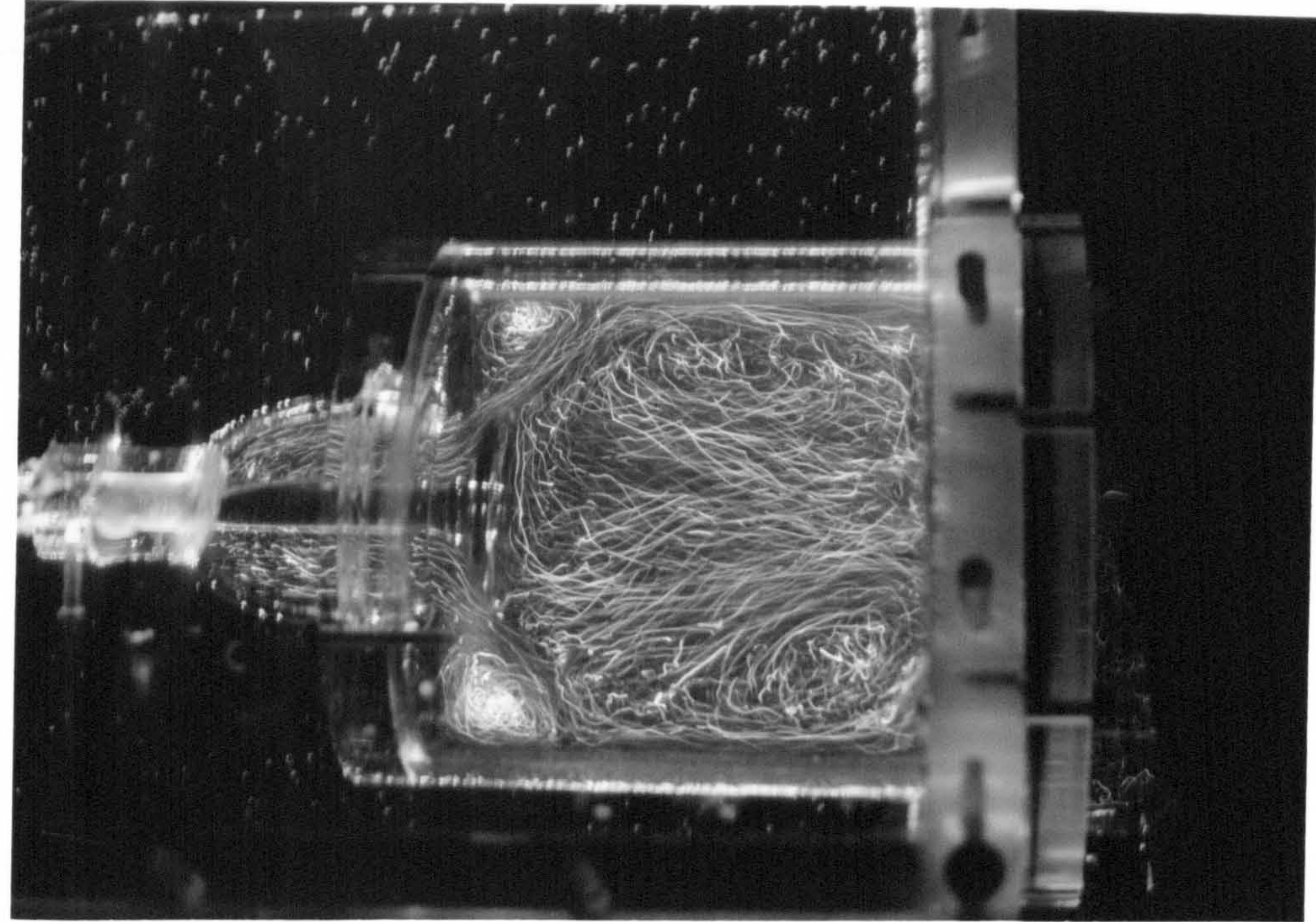


Plane B

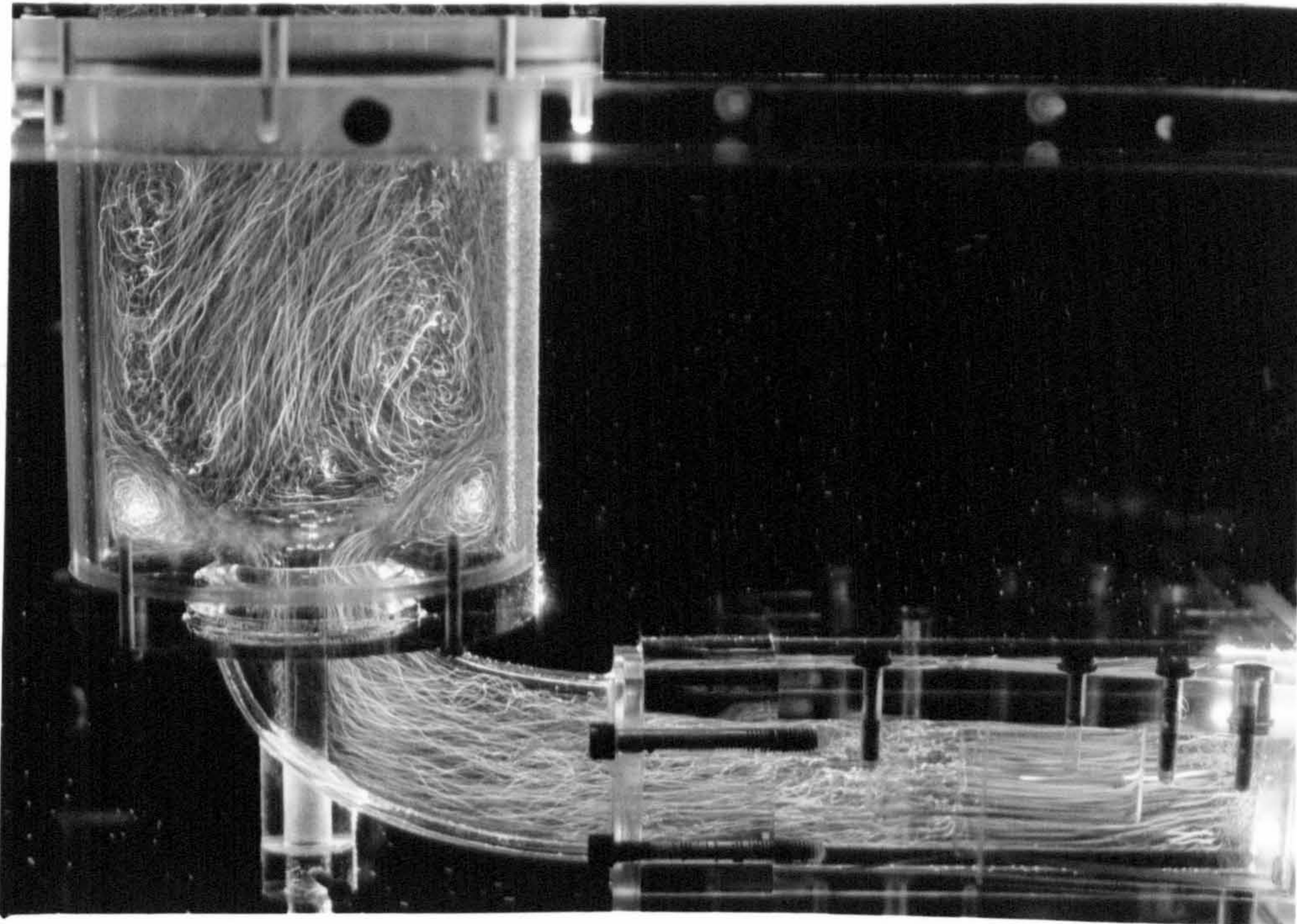


Plane A

Plate 4.3 Photographs of Planes A and B for variable geometry ramp model at $\theta = 130^\circ$

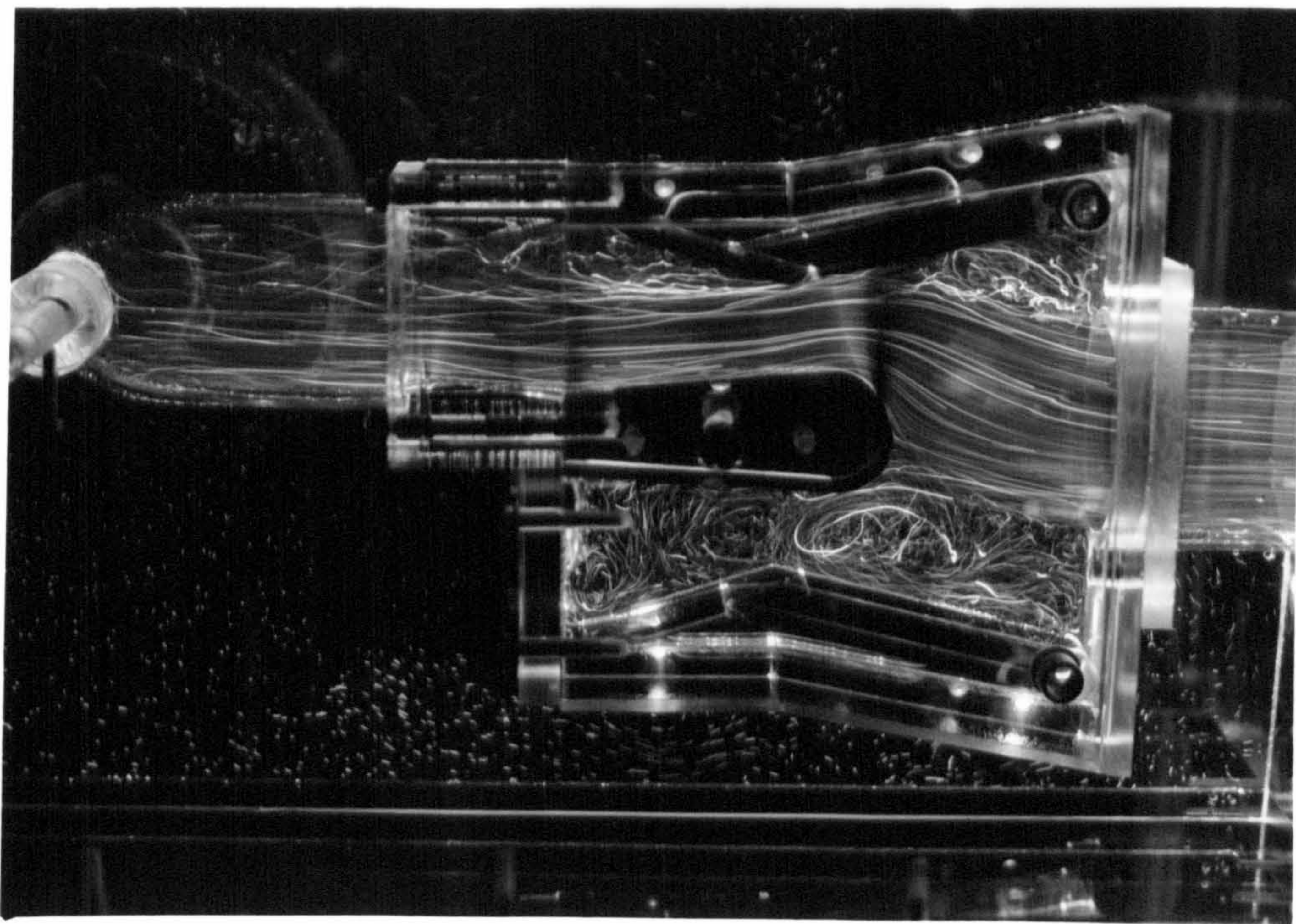


Plane D

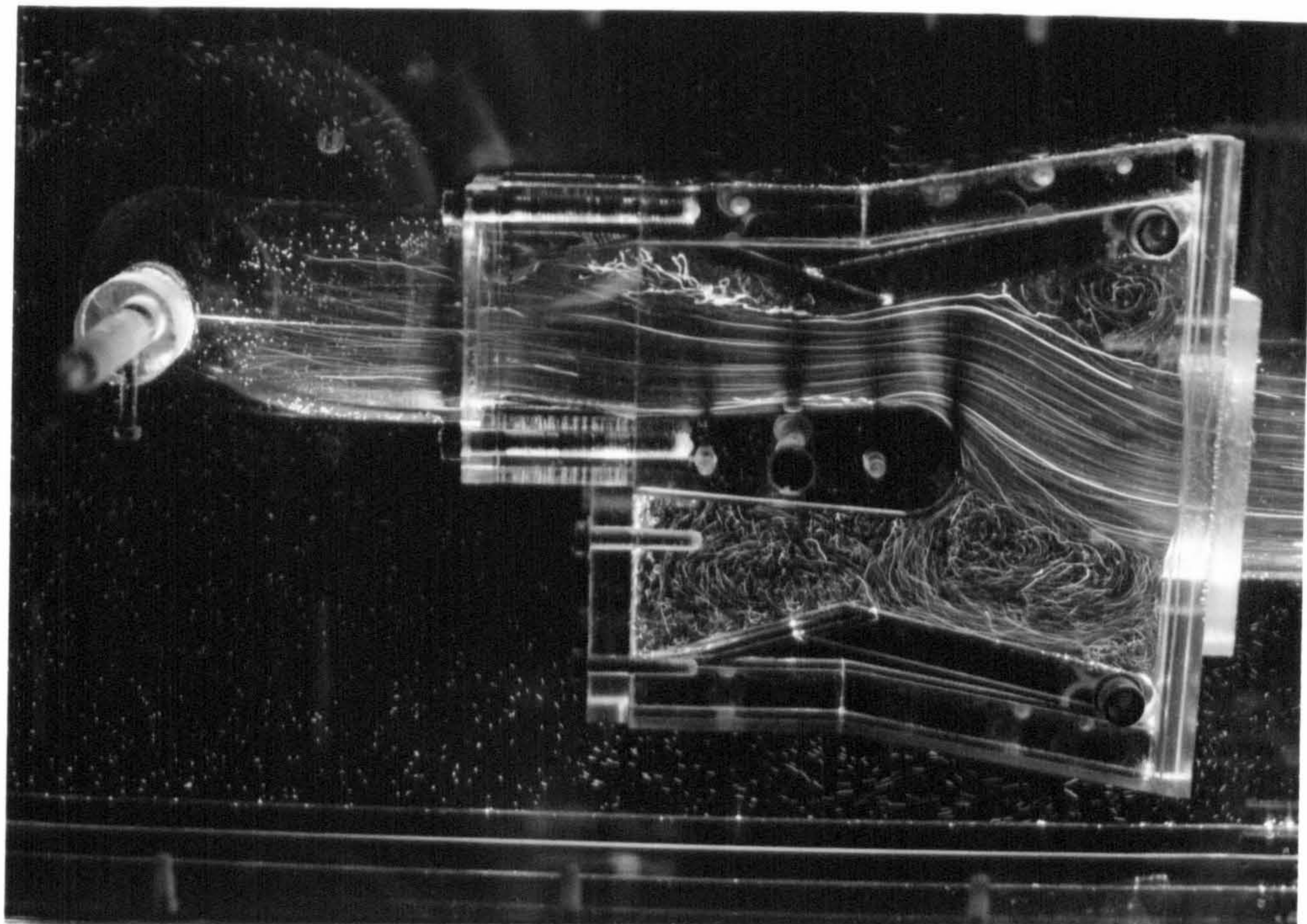


Plane C

Plate 4.4 Photographs of Planes C and D for variable geometry ramp throttling model at $\theta = 130^\circ$

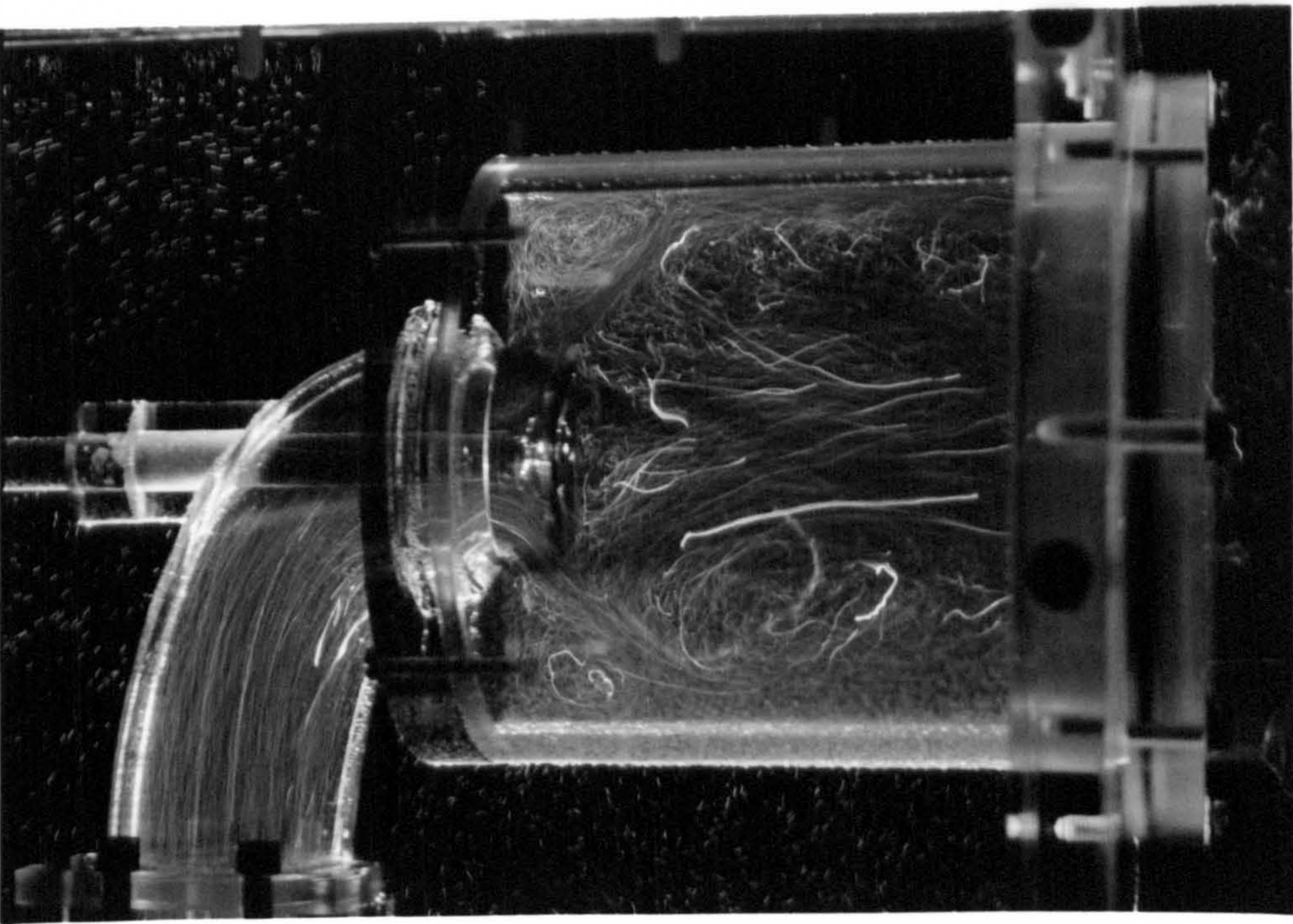


Plane A

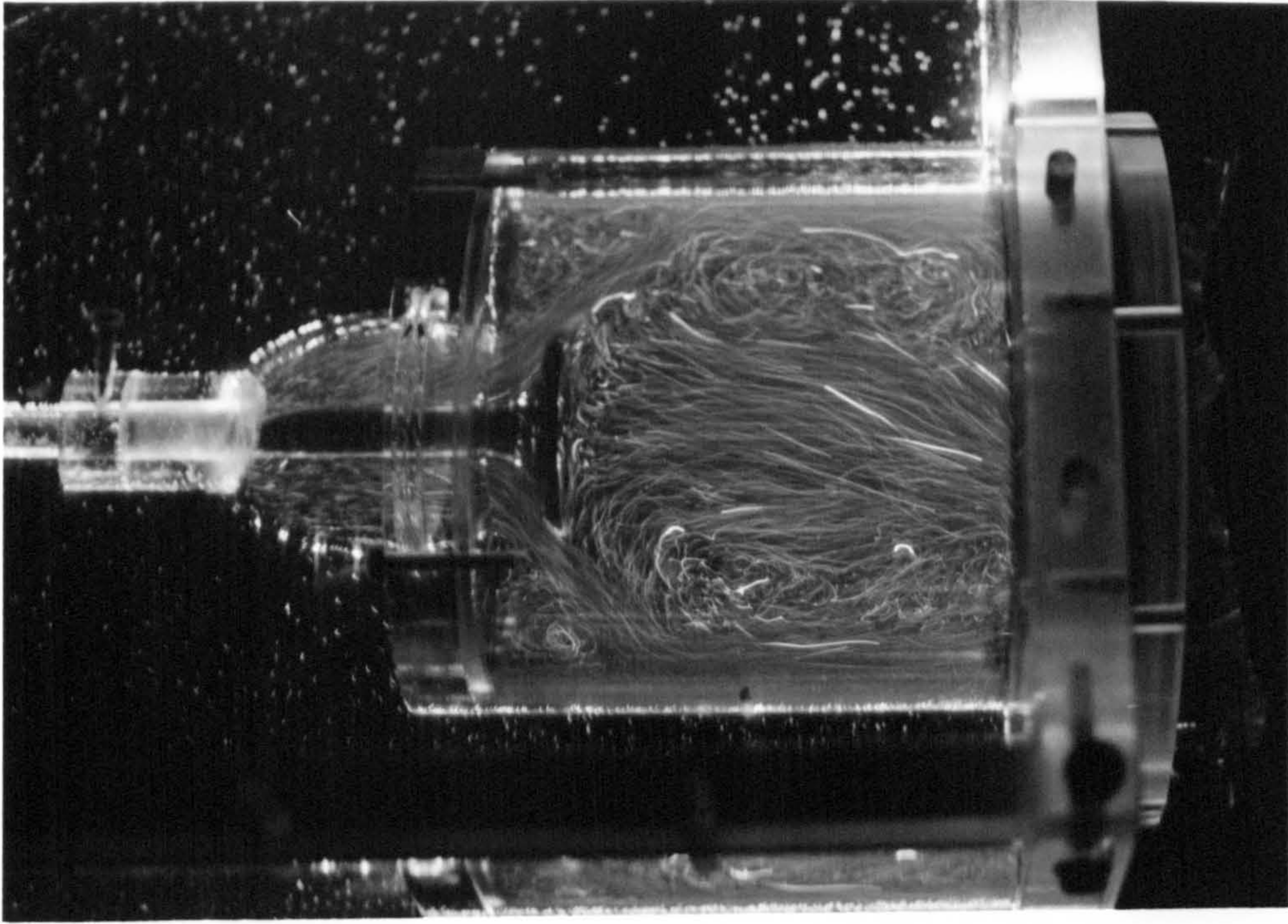


Plane B

Plate 4.5 Photographs of Planes A and B for variable geometry ramp throttling model at $\theta = 150^\circ$

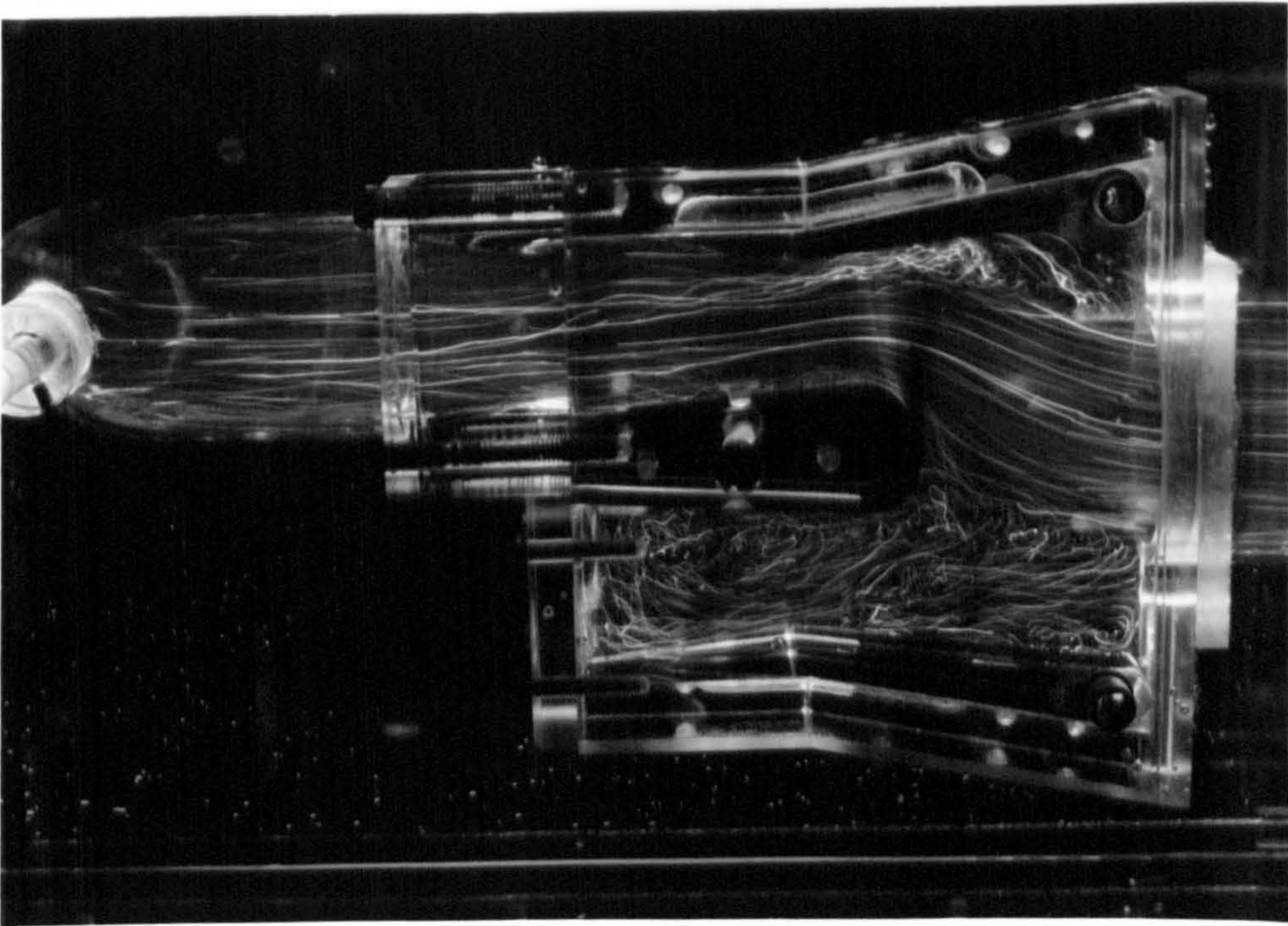


Plane C

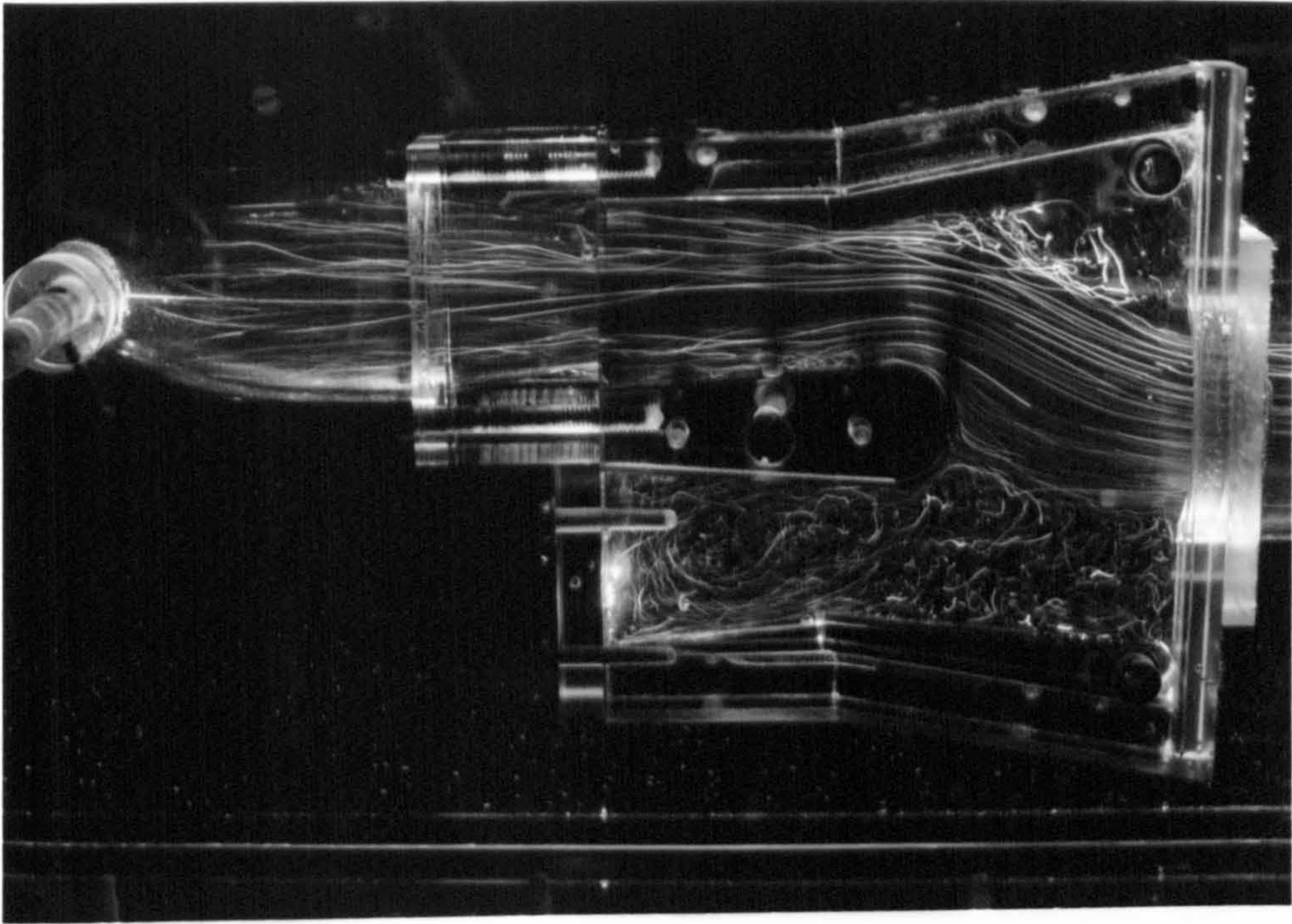


Plane D

Plate 4.6 Photographs of Planes C and D for variable geometry ramp throttling model
at $\theta = 150^\circ$

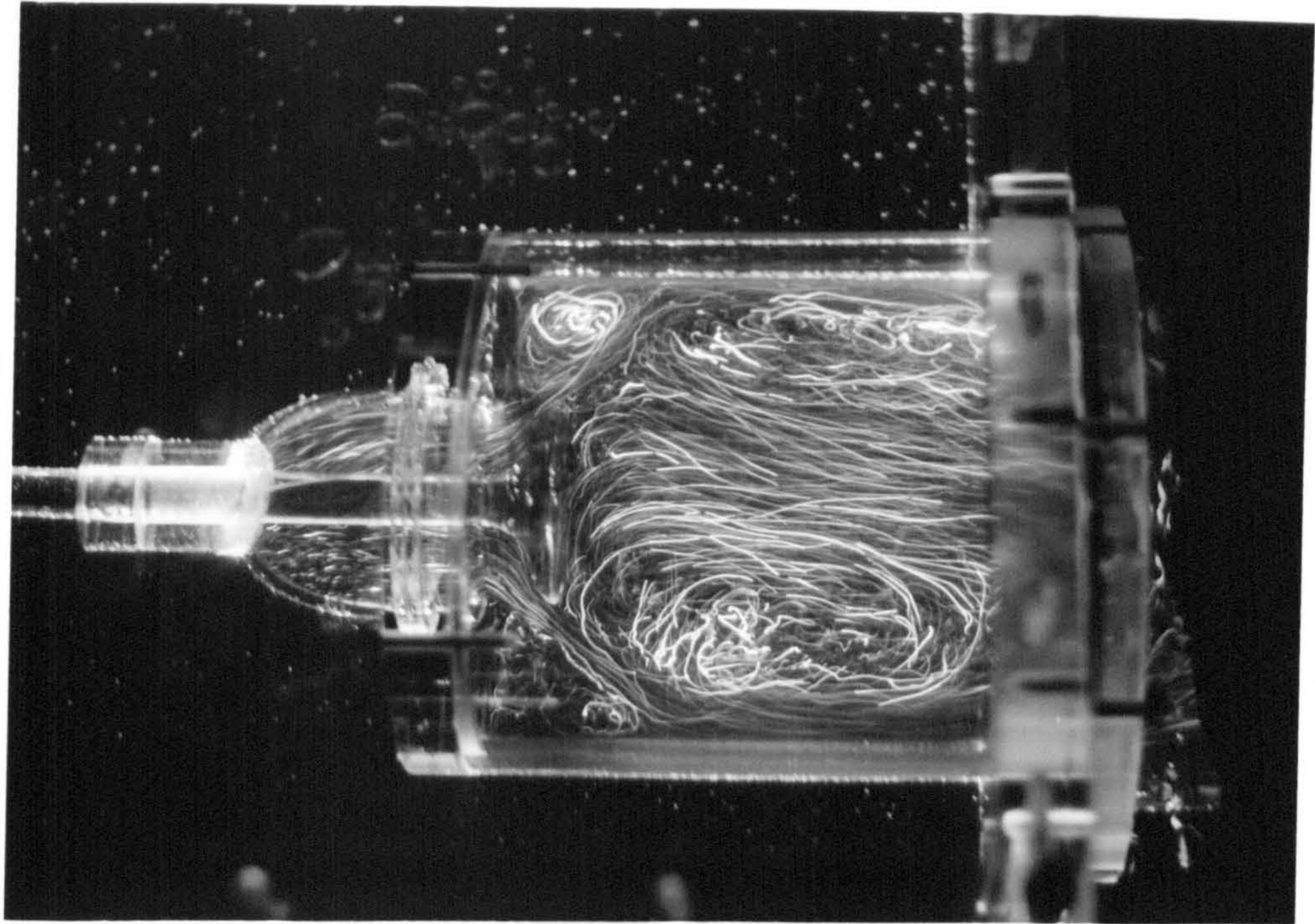


Plane A

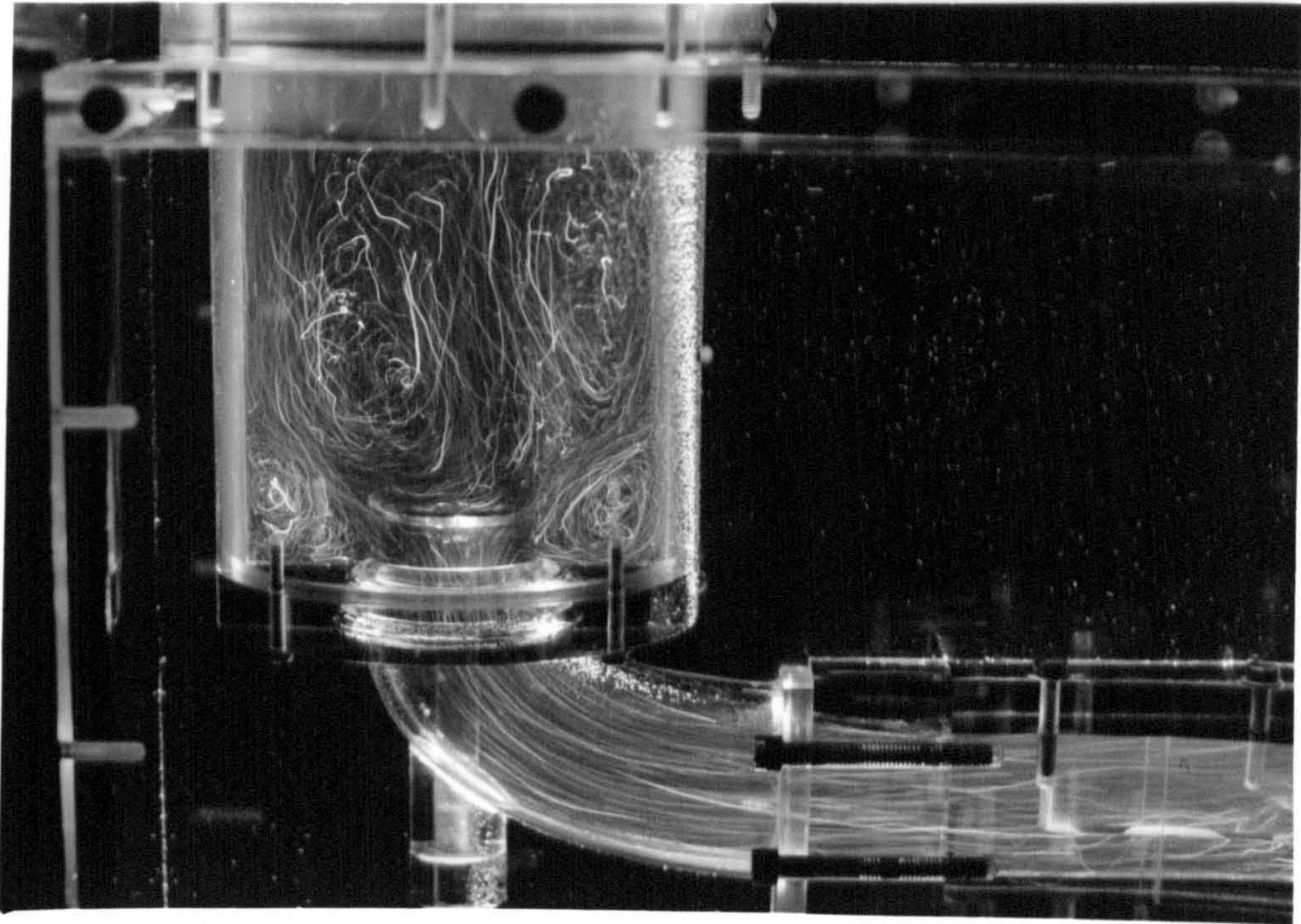


Plane B

Plate 4.7 Photographs of Planes A and B for variable geometry ramp throttling model at WOT

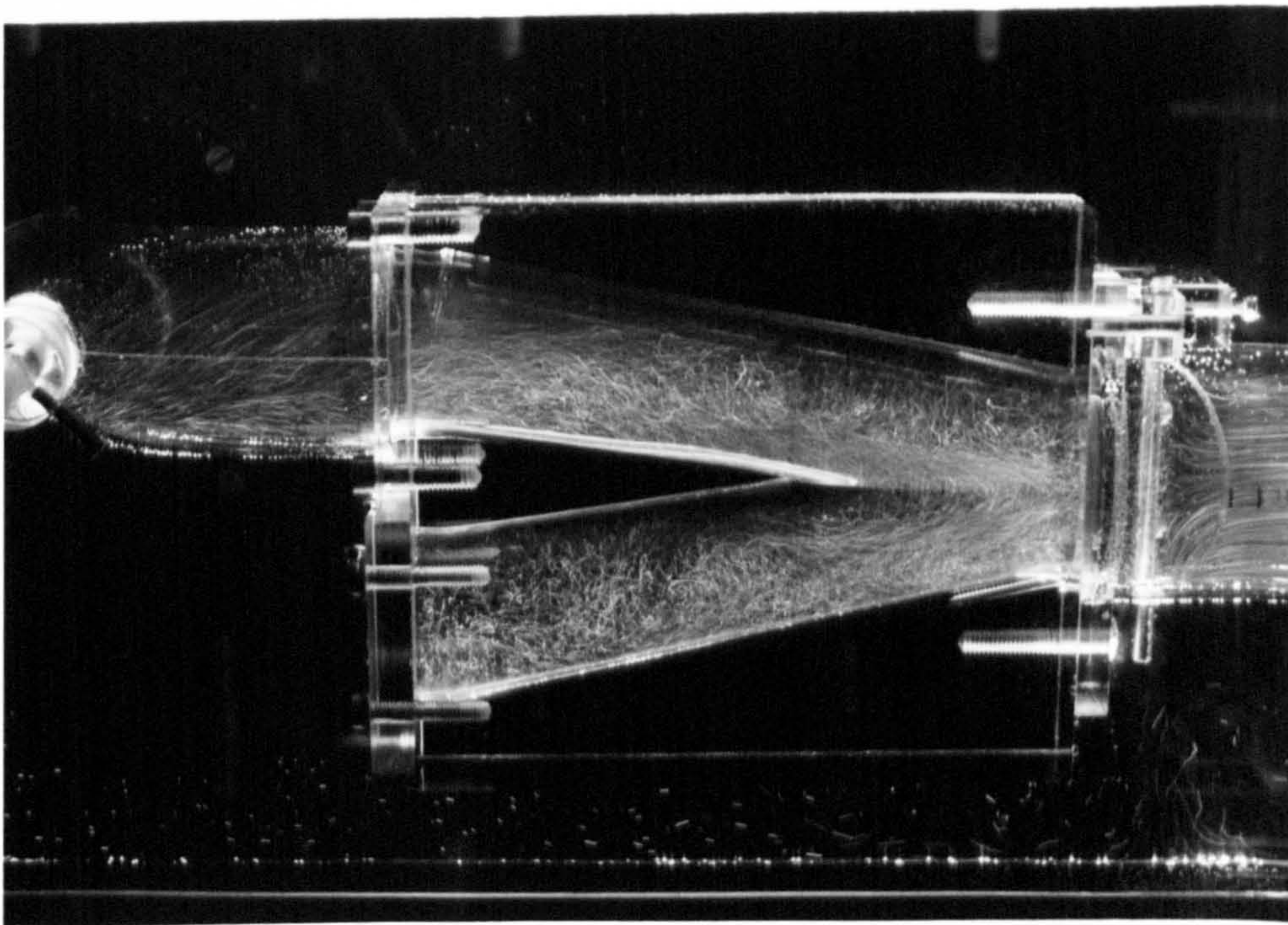


Plane D

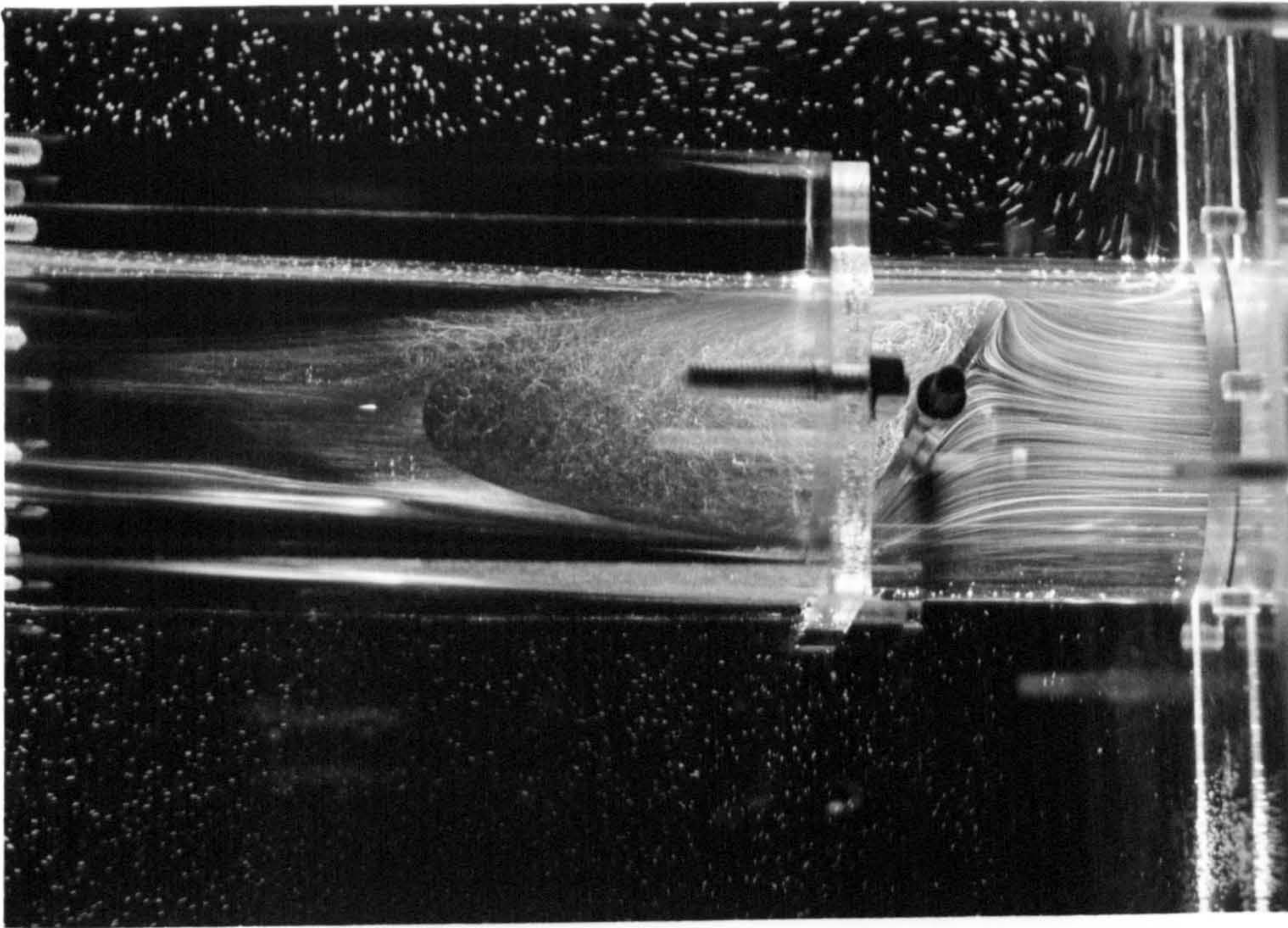


Plane C

Plate 4.8 Photographs of Planes C and D for variable geometry ramp throttling model at WOT

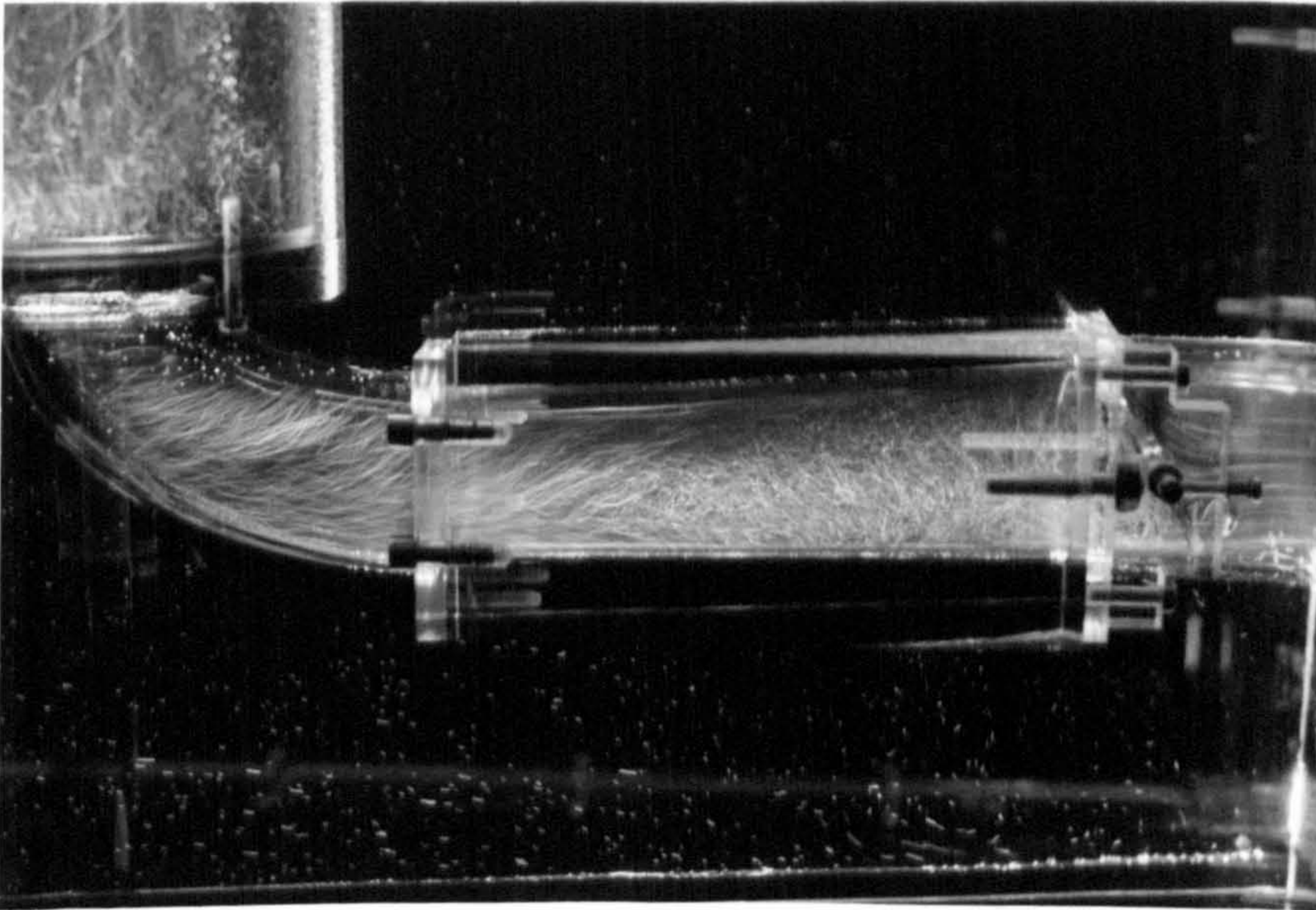


Plane A'

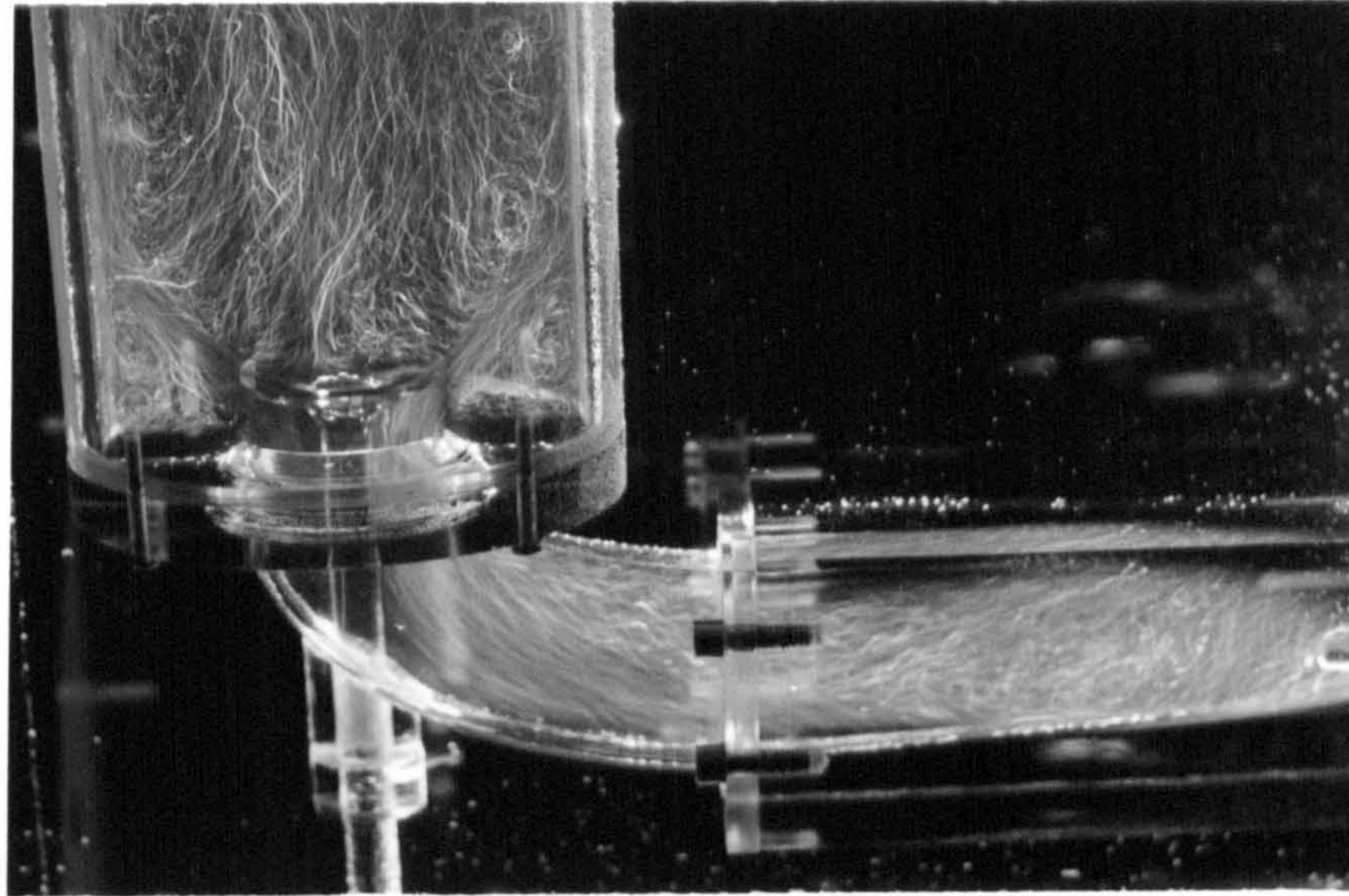


Plane B'

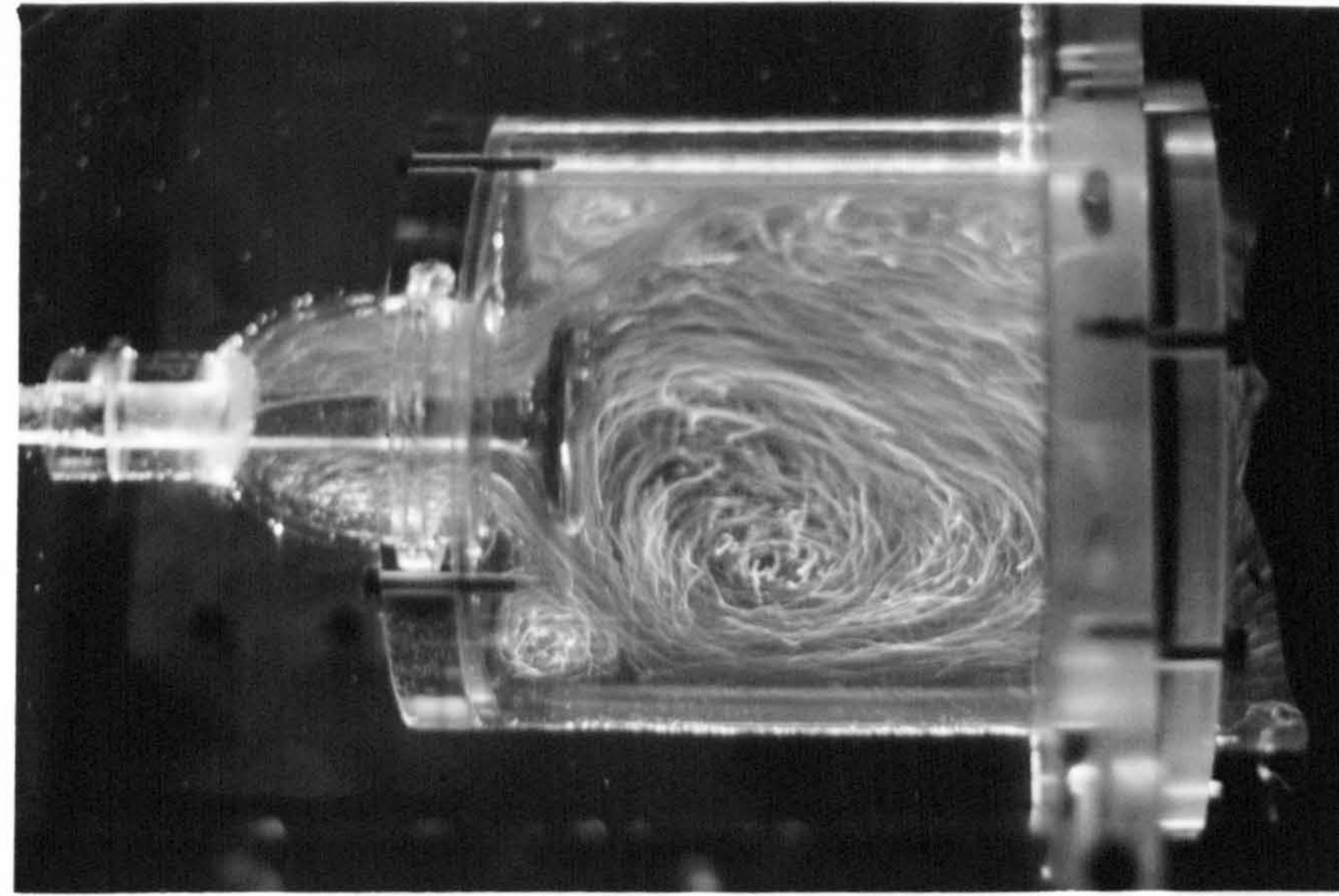
Plate 4.9 Photographs of Planes A' and B' for butterfly valve model at $\beta = 30^\circ$



Plane C'

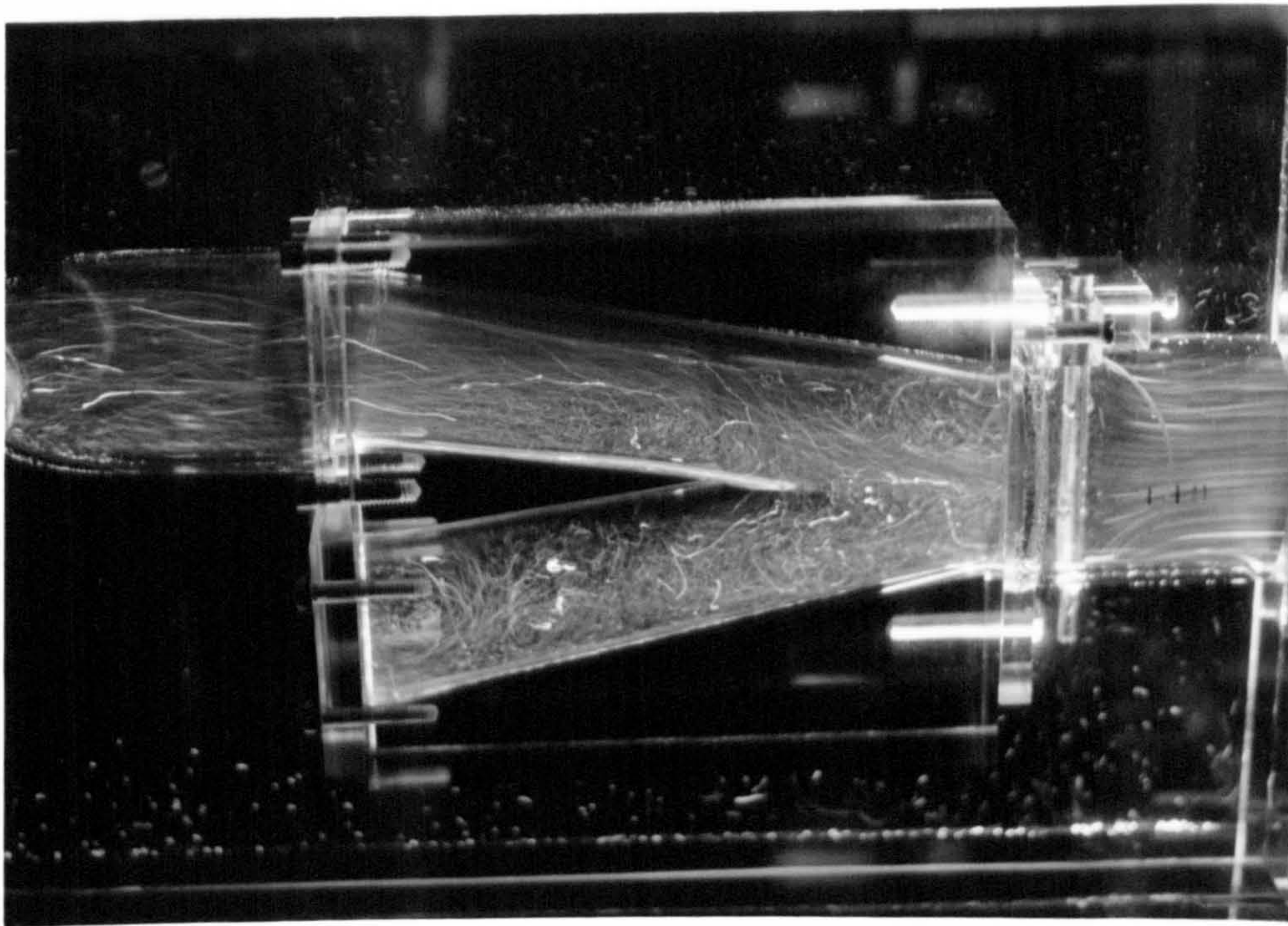


Plane D'

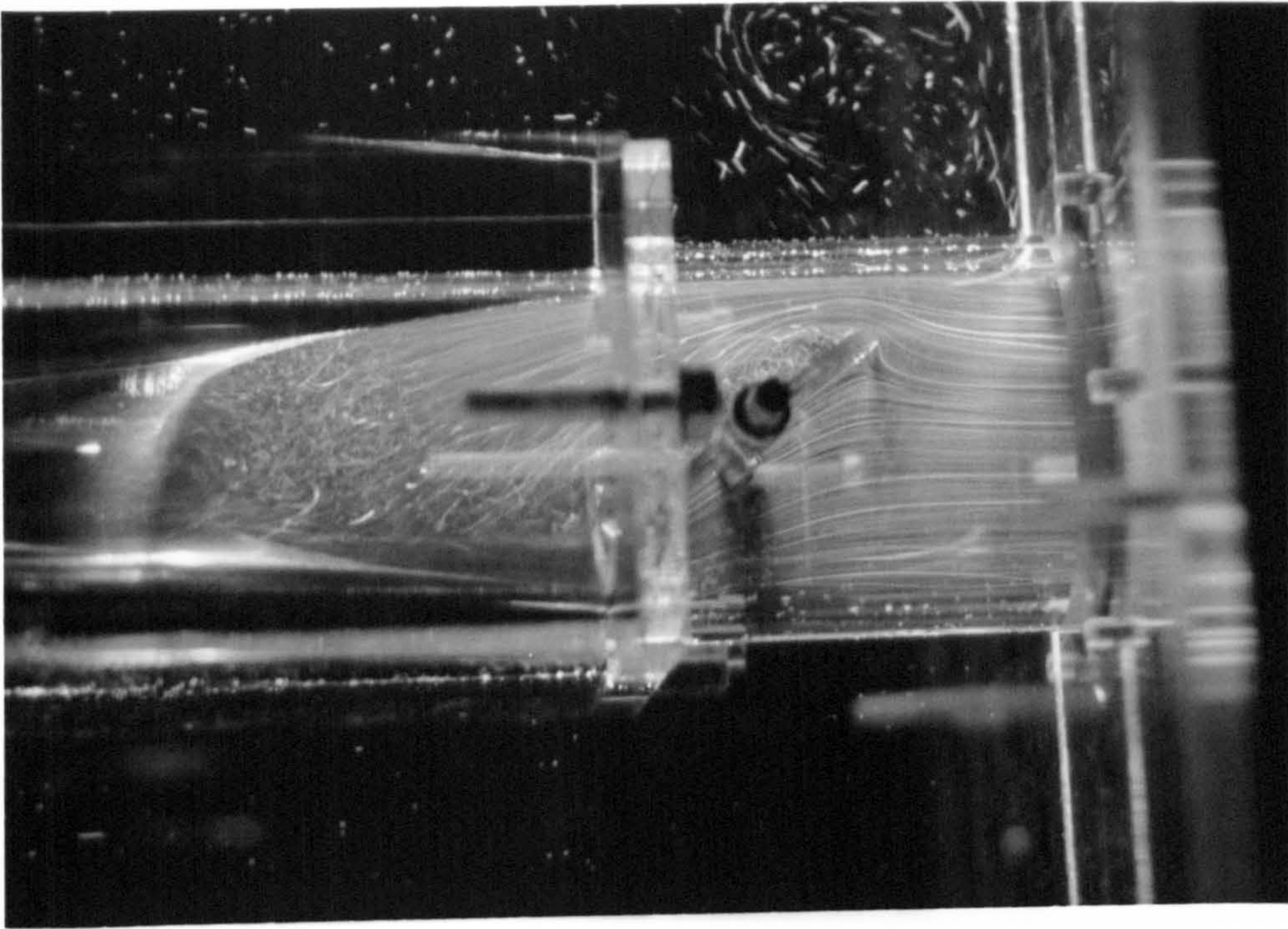


Plane E'

Plate 4.10 Photographs of Planes C', D' and E' for butterfly valve model at $\beta = 30^\circ$

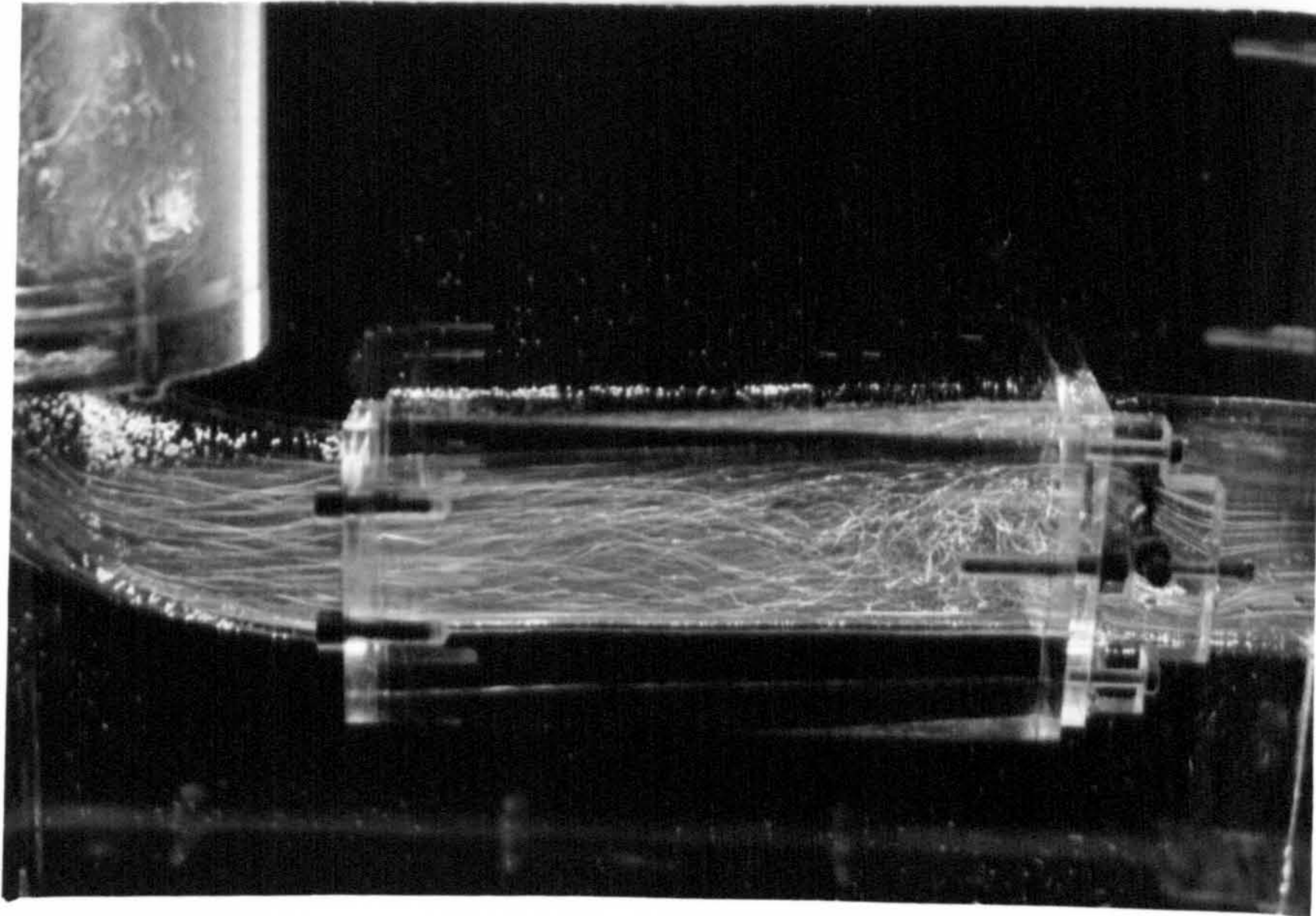


Plane A'

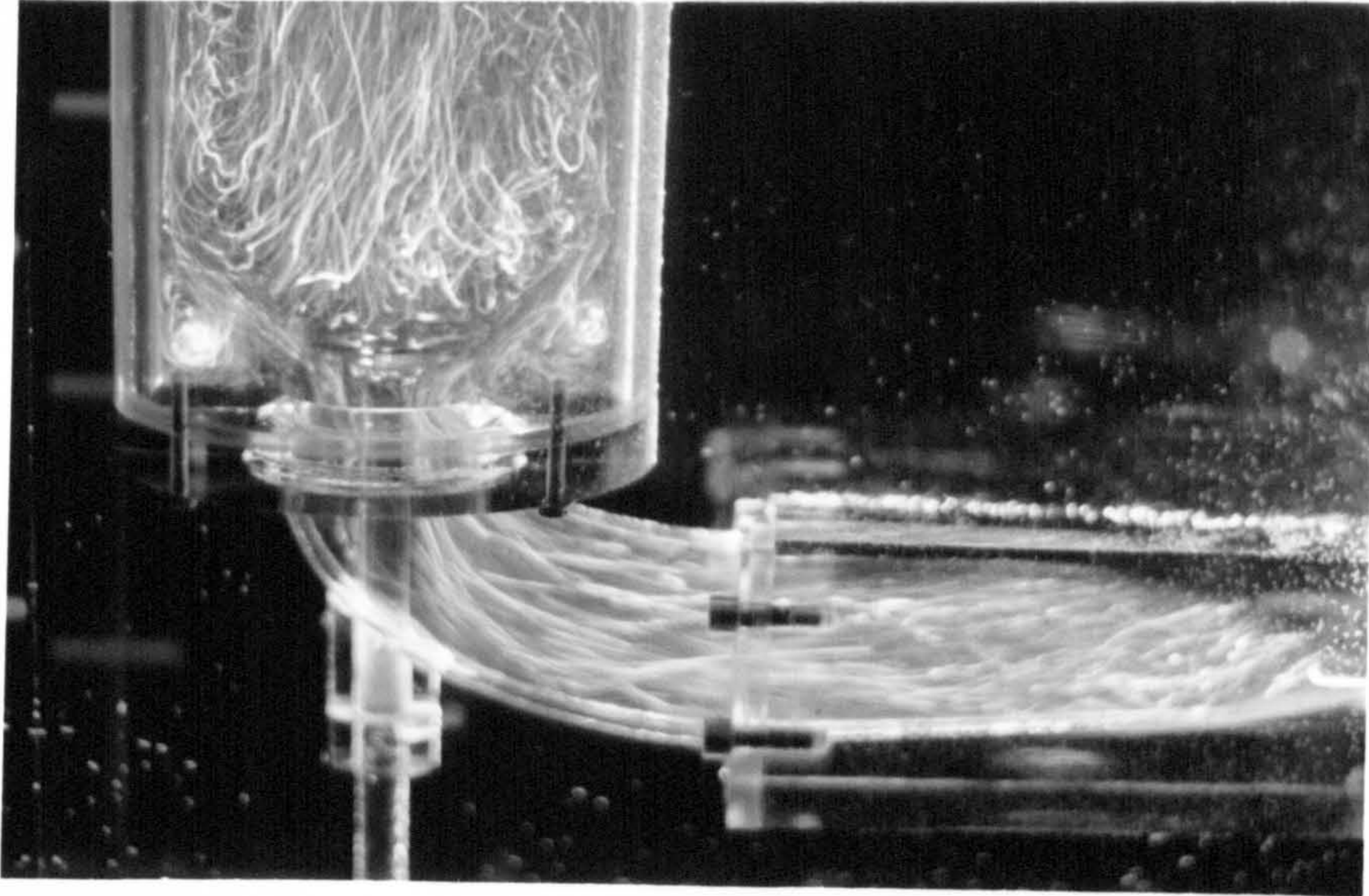


Plane B'

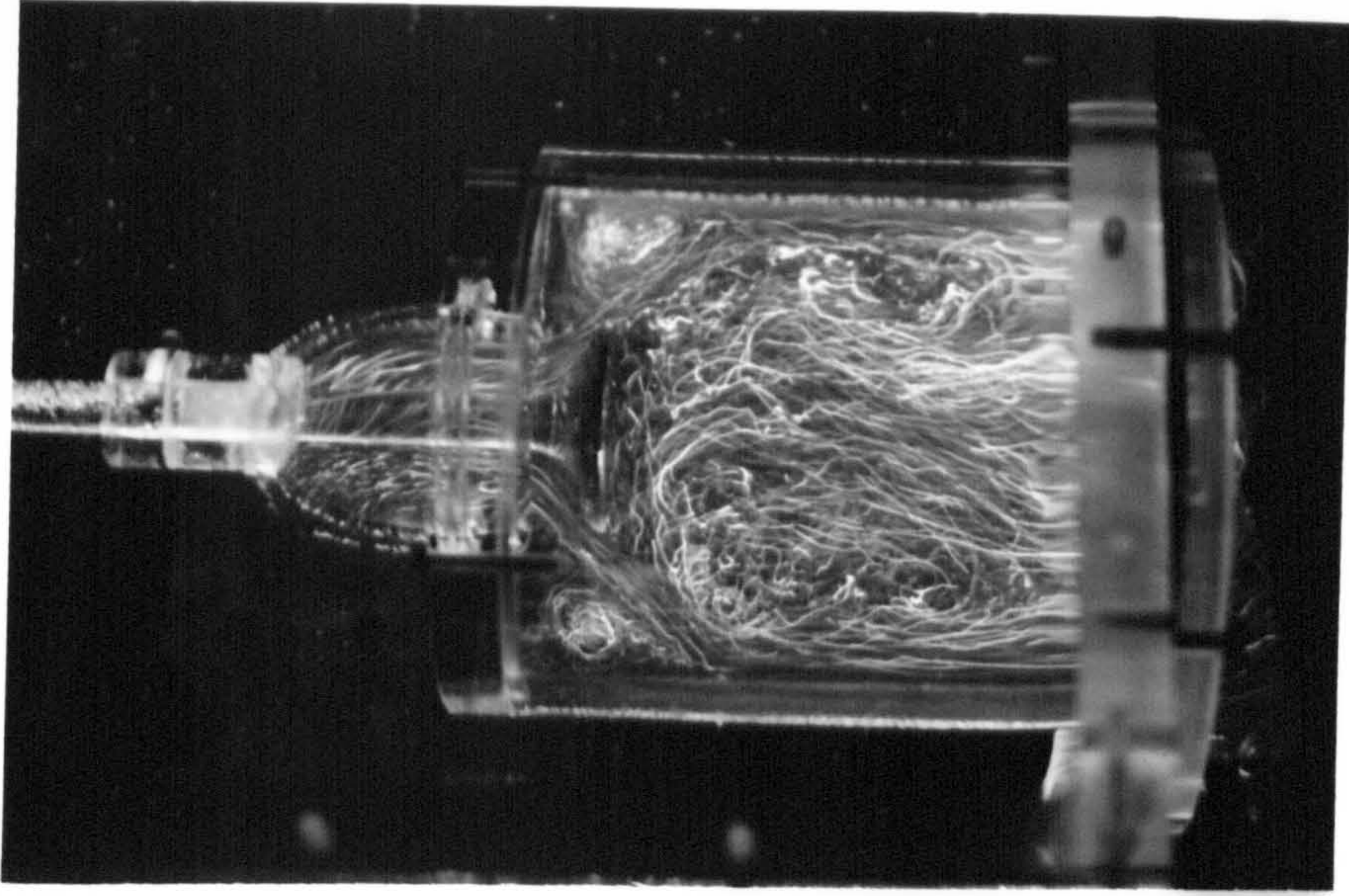
Plate 4.11 Photographs of Planes A' and B' for butterfly valve model at $\beta = 50^\circ$



Plane C'

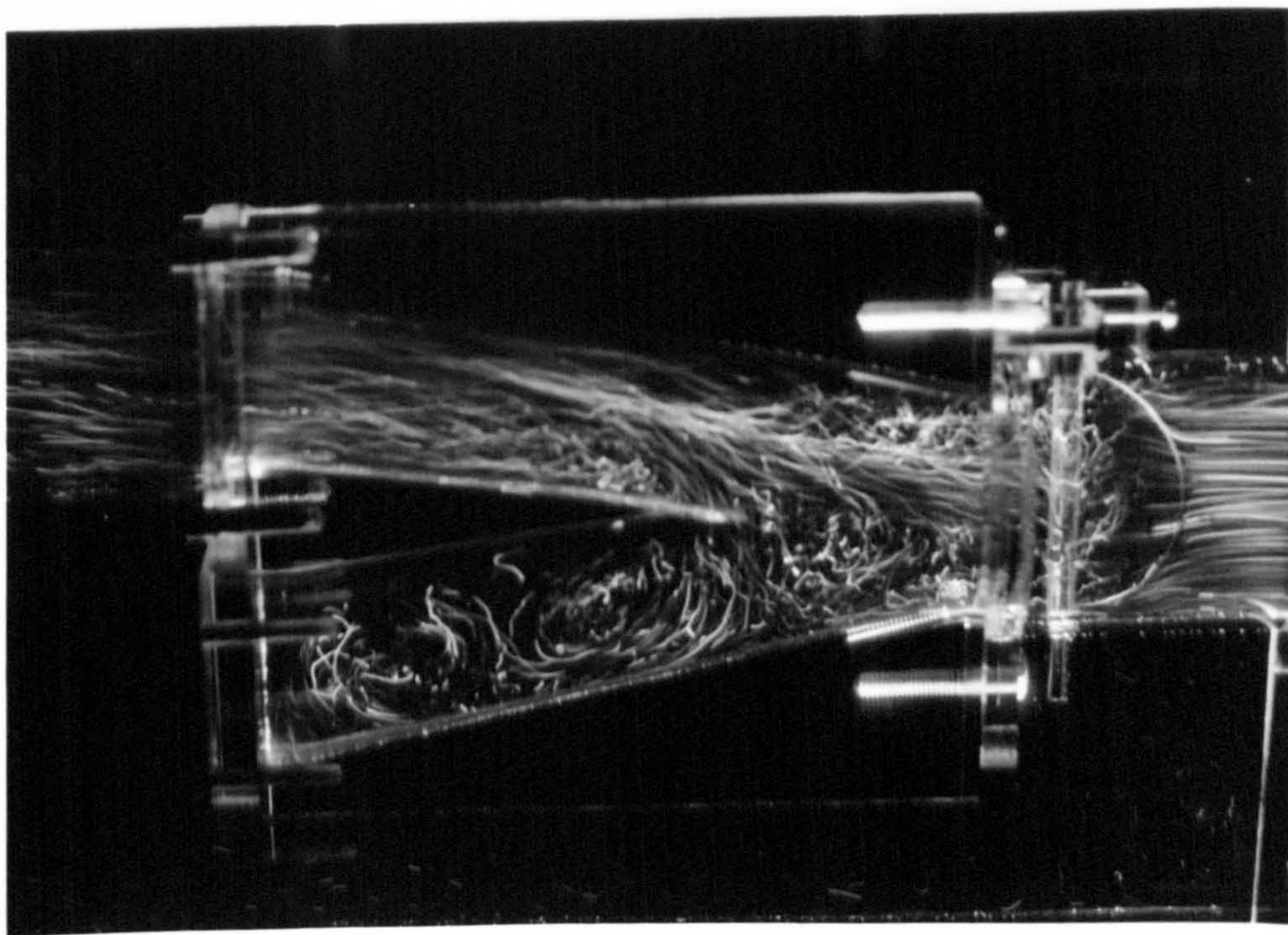


Plane D'

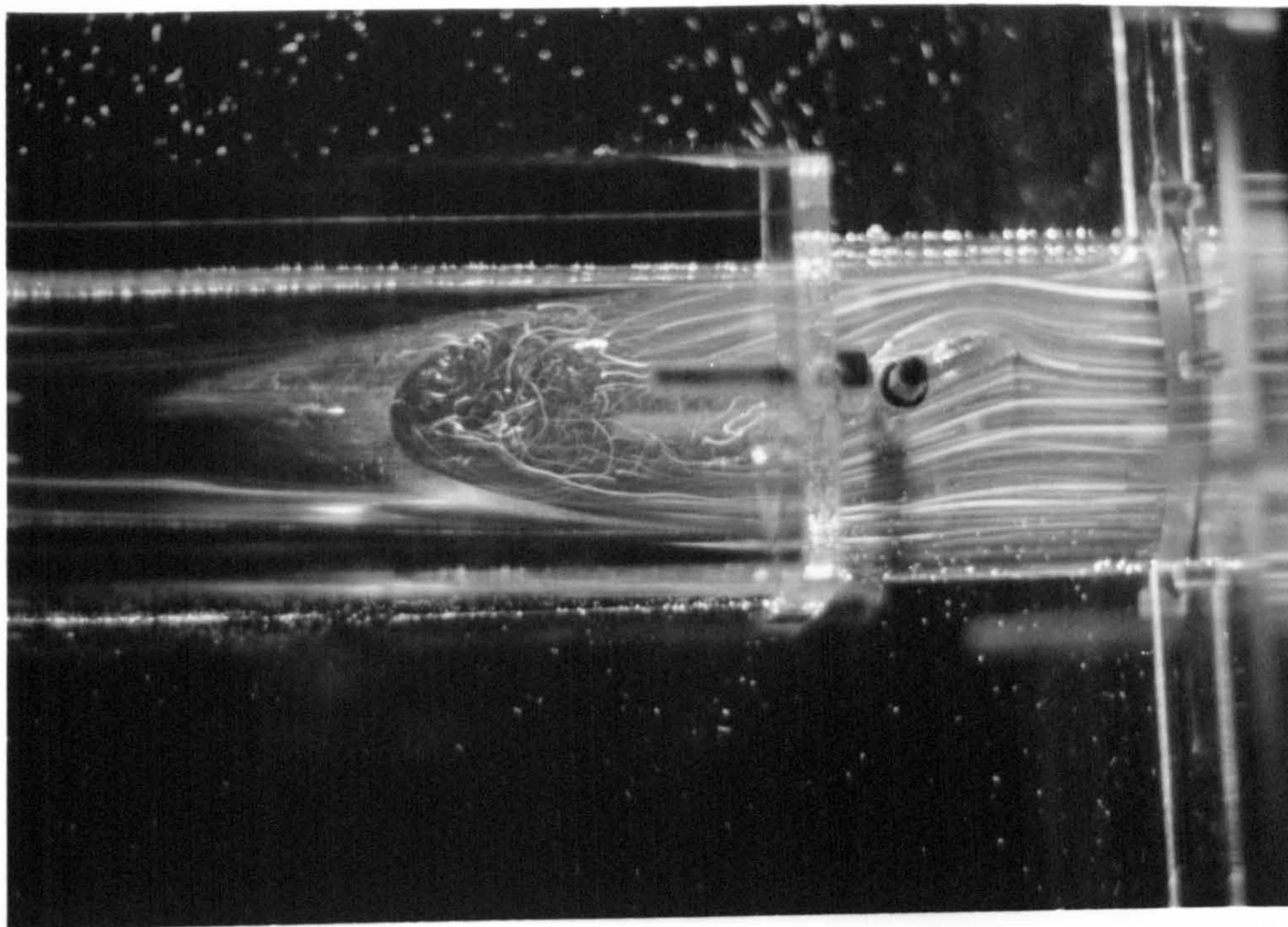


Plane E'

Plate 4.12 Photographs of Planes C', D' and E' for butterfly valve model at $\beta = 50^\circ$

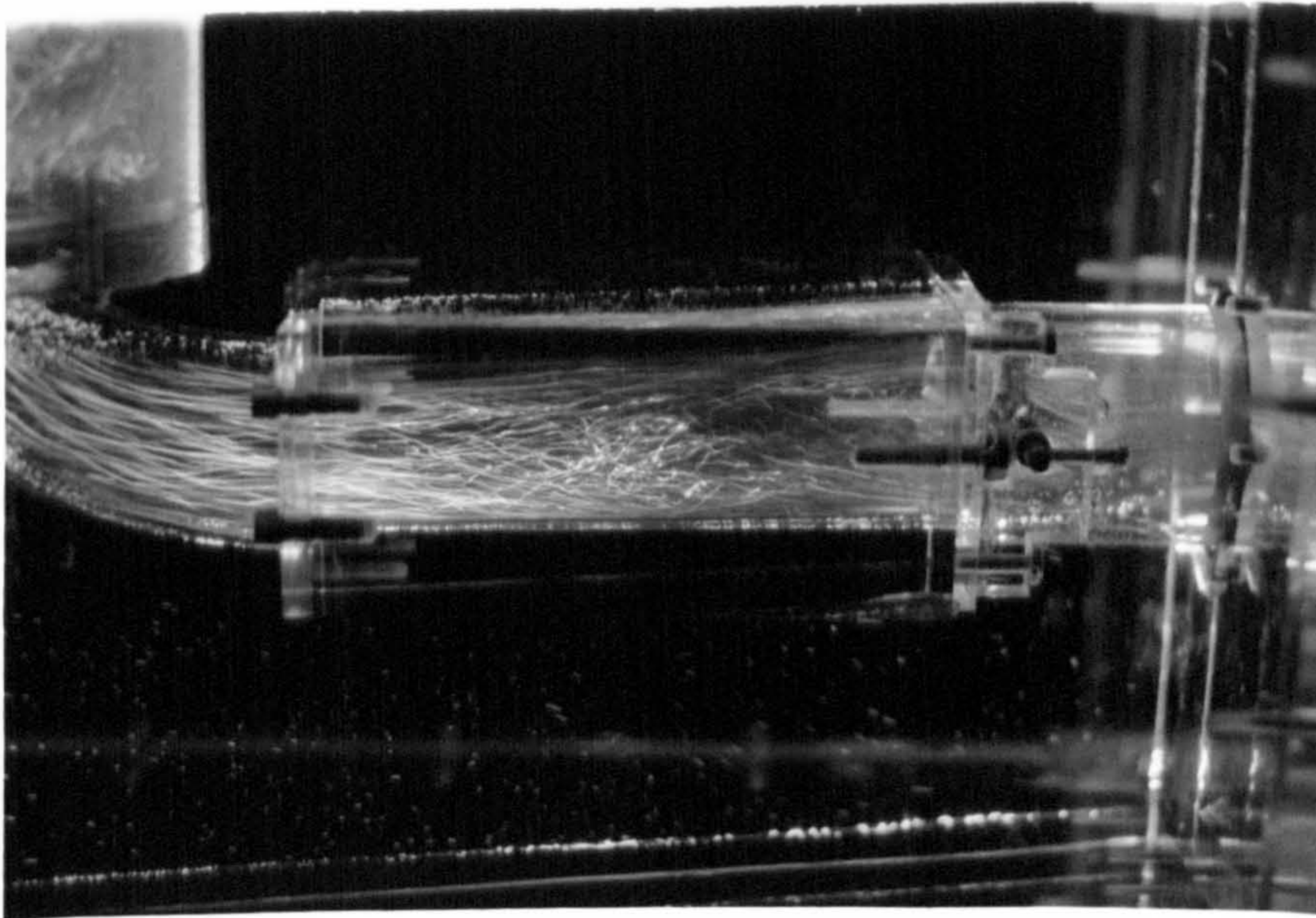


Plane A'

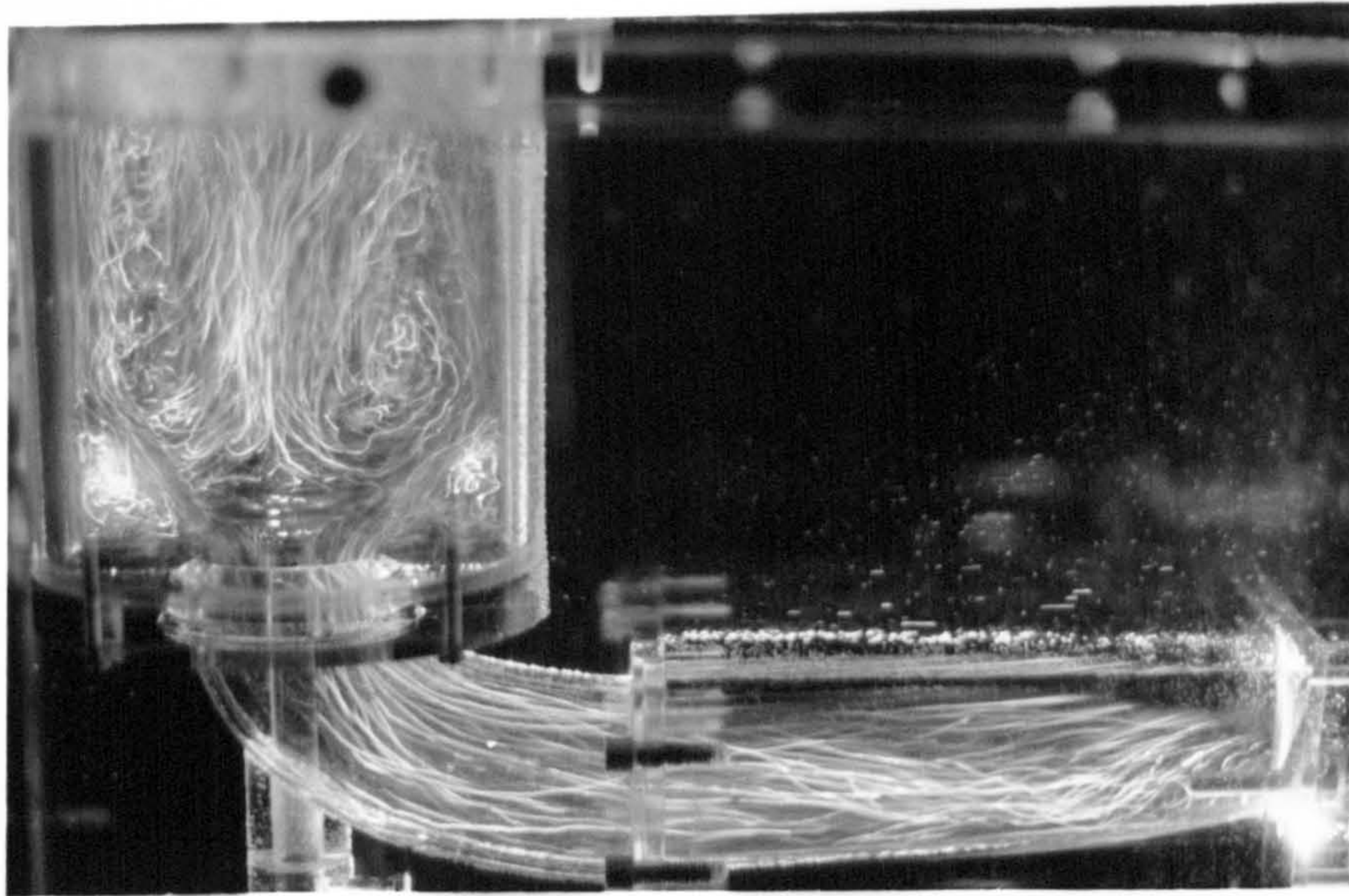


Plane B'

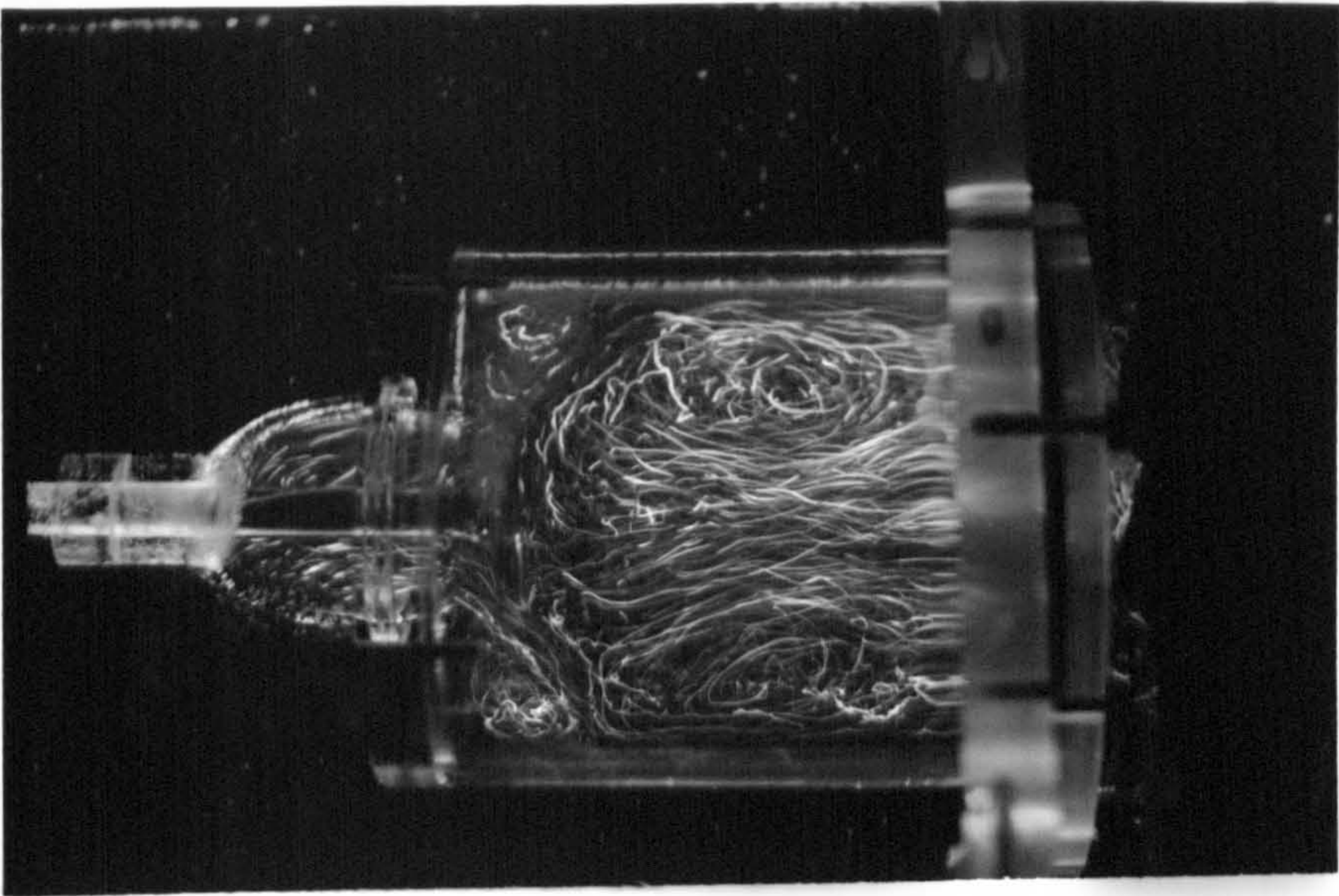
plate 4.13 Photographs of Planes A' and B' for butterfly valve model at $\beta = 65^\circ$



Plane C'

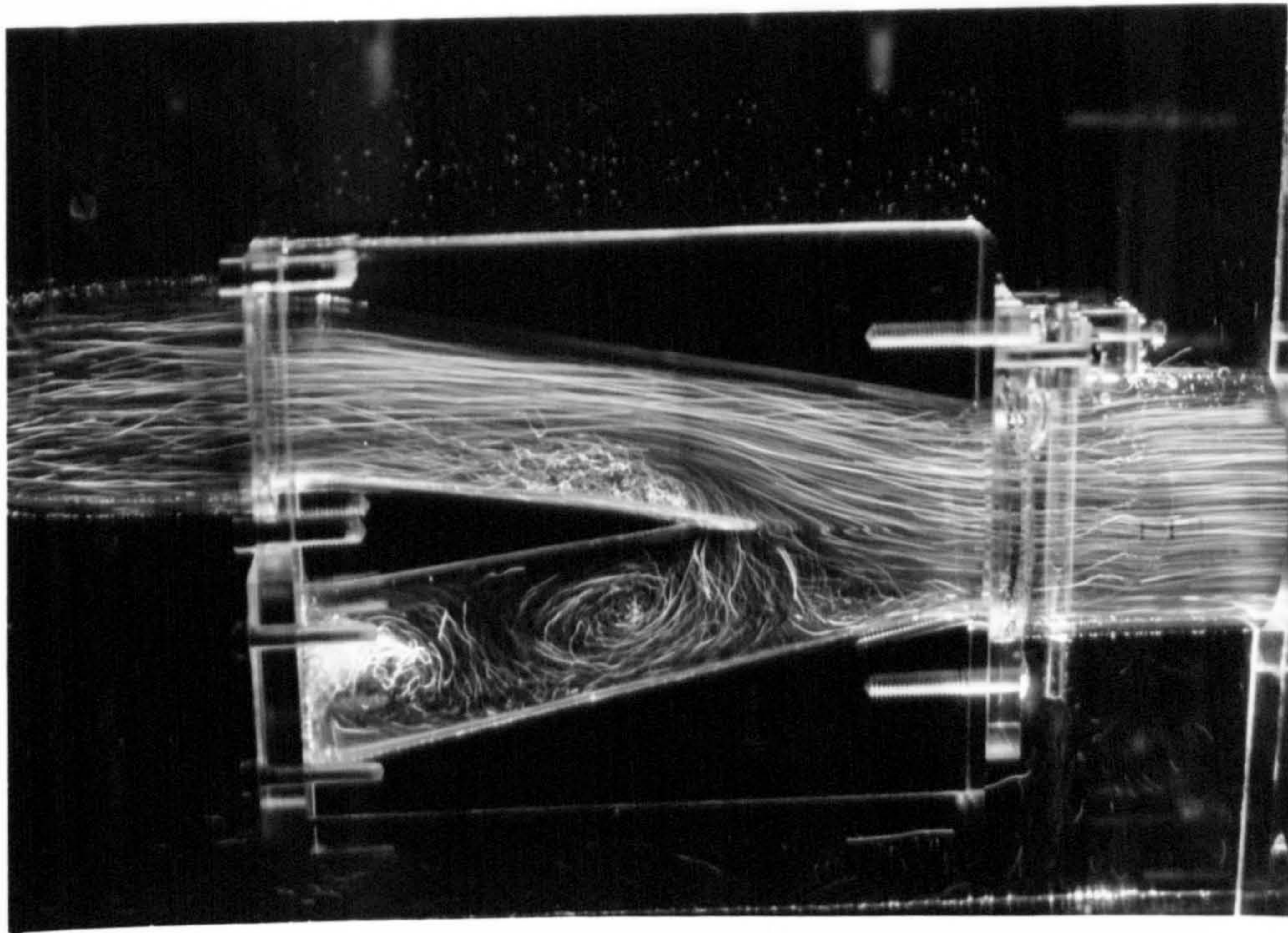


Plane D'

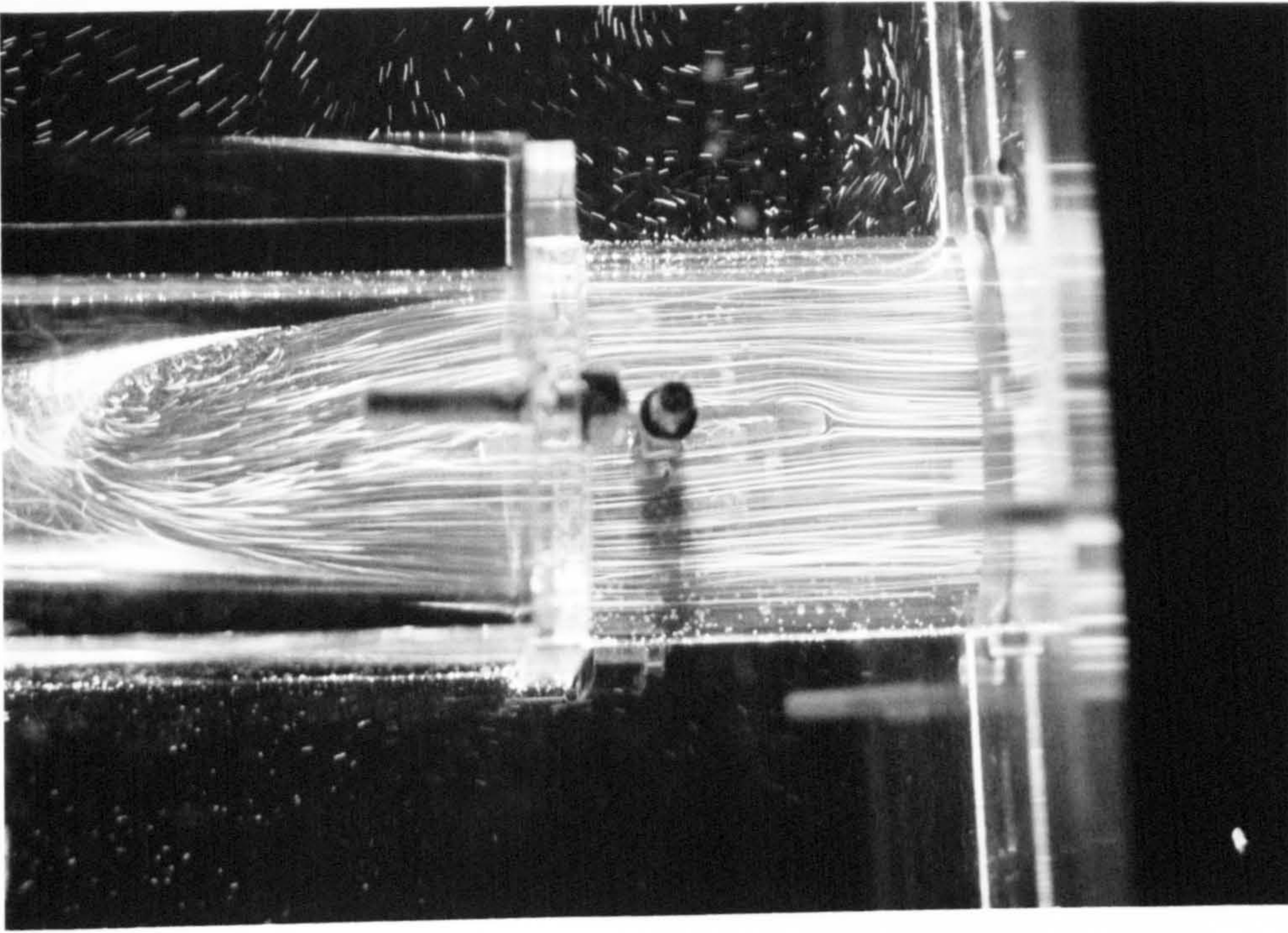


Plane E'

Plate 4.14 Photographs of Planes C', D' and E' for butterfly valve model at $\beta = 65^\circ$

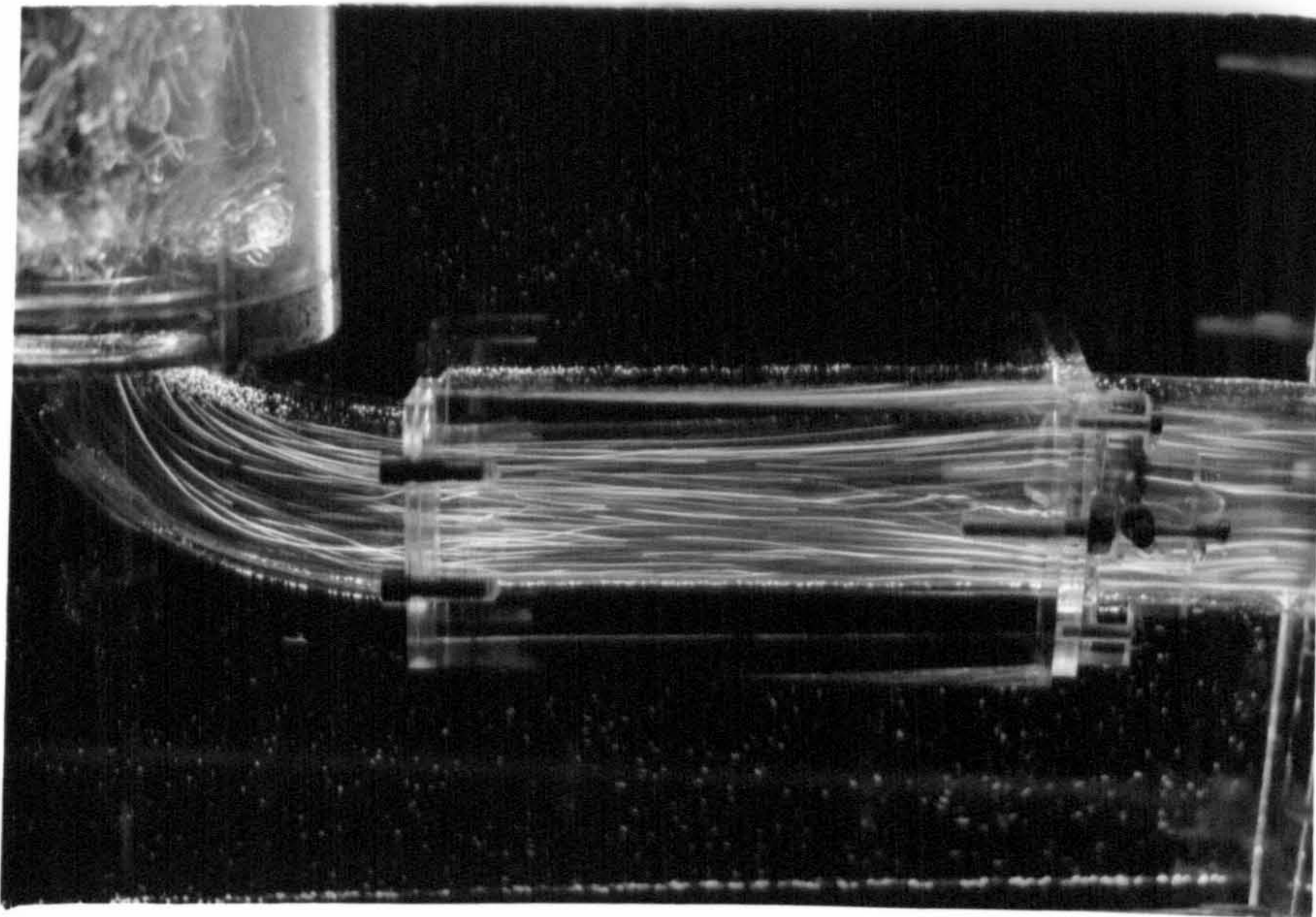


Plane A'

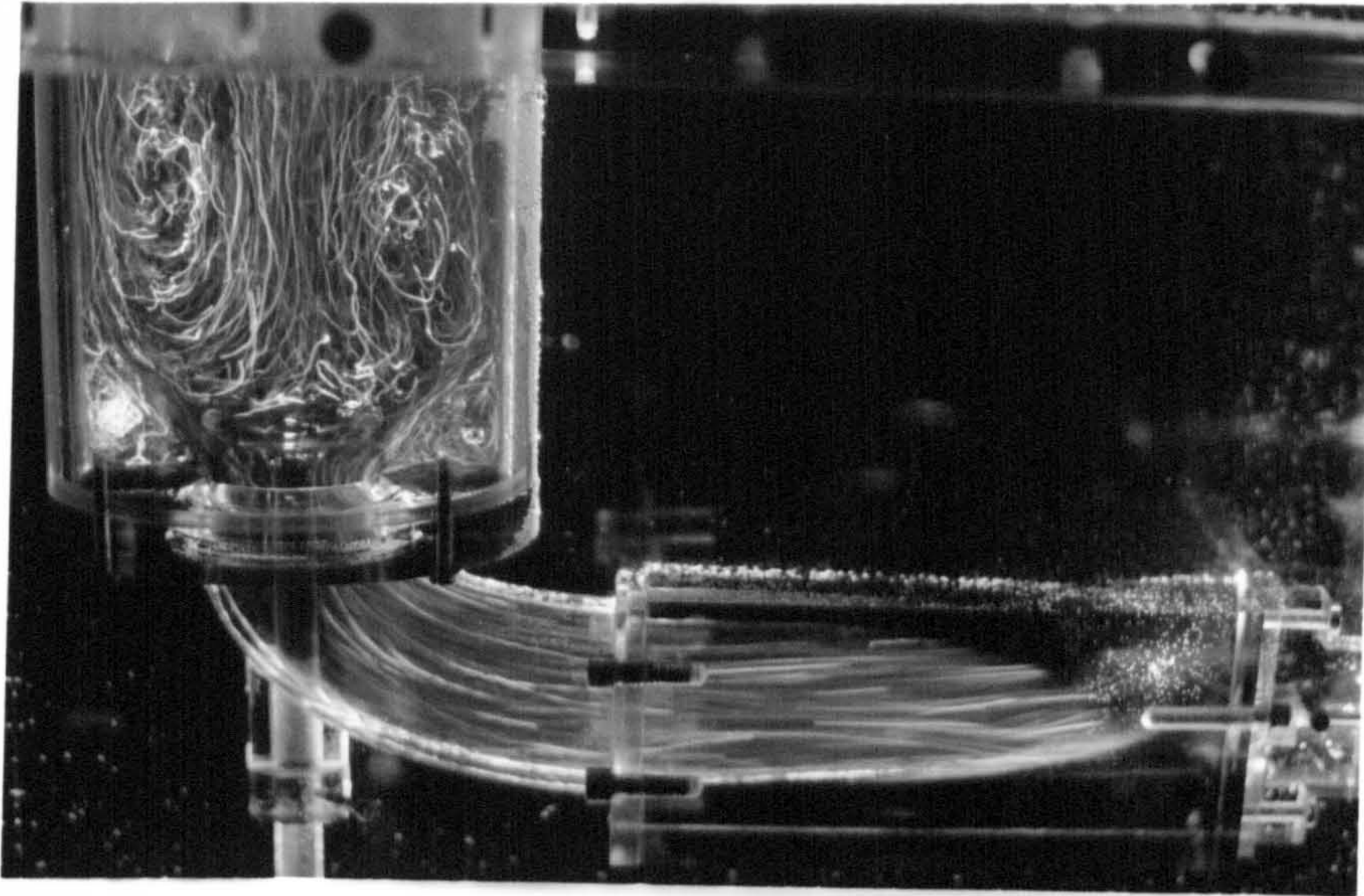


Plane B'

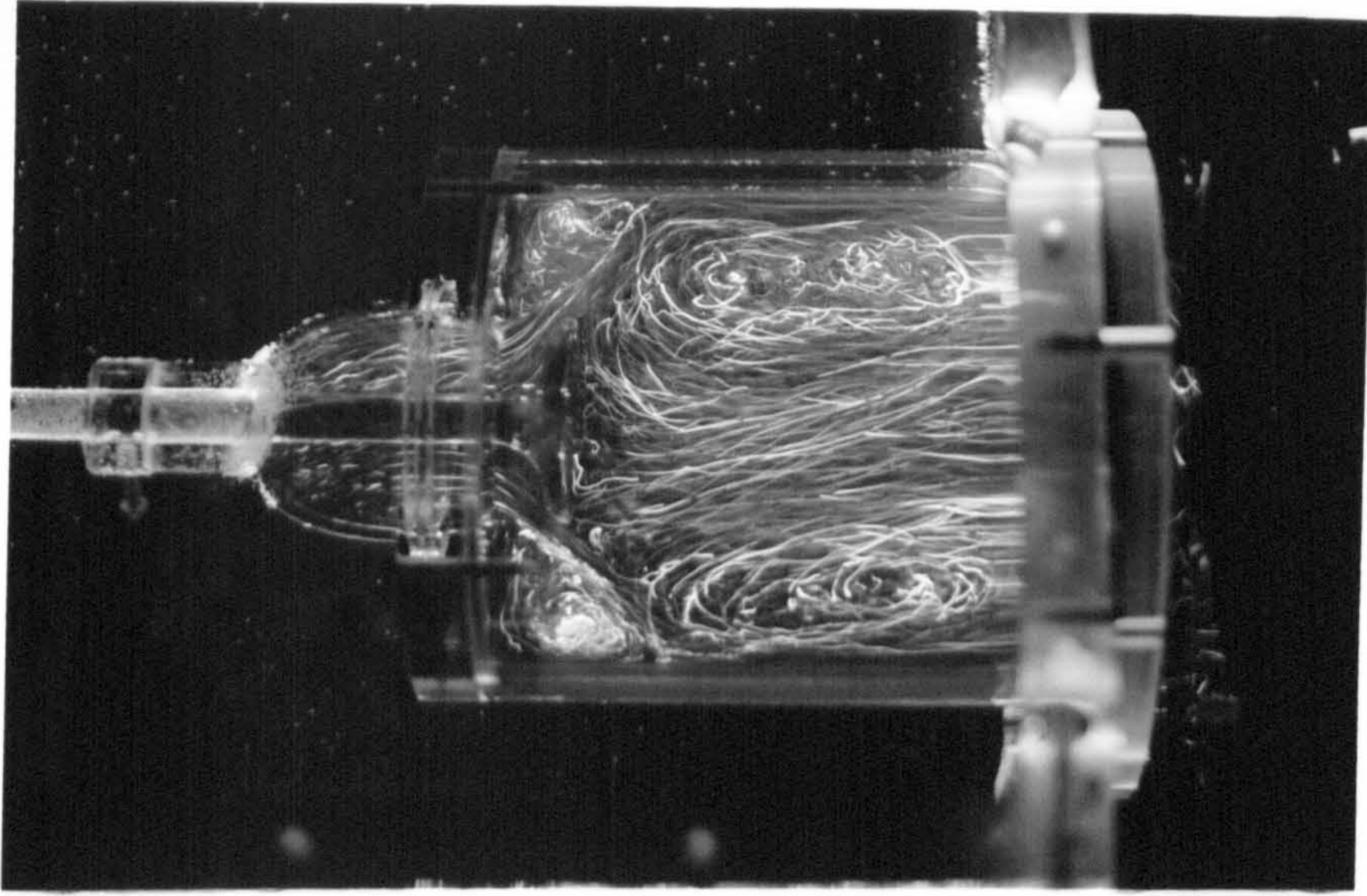
Plate 4.15 Photographs of Planes A' and B' for butterfly valve model at WOT



Plane C'



Plane D'



Plane E'

Plate 4.16 Photographs of Planes C', D' and E' for butterfly valve model at WOT

CHAPTER 5

STEADY FLOW VELOCITY MEASUREMENTS OF THE INDUCTION FLOW

5.1 Introduction

From the previous chapter the flow field visualization tests, though informative, provided mainly qualitative information about the flow field structure and the extent and strength of mixing and recirculation zones in the butterfly valve and variable geometry ramp systems. However, to achieve the objective of the present studies we require quantitative data, and in particular velocity measurements to provide both the means of direct comparison between the two alternative devices and also to provide a means by which the computer simulated models could be validated. Of these two, the latter was of more direct interest in the present study.

In this chapter, the largely qualitative data obtained from the water analogy tests reported in the previous chapter are complemented by hot wire velocity measurements at particular stations in the Perspex models, with air replacing the water as the flow medium.

The Perspex models used are identical to the ones used in flow visualization tests, with the exception of those minor alterations made to provide access ports at appropriate stations for the hot wire probes.

5.2 Measurement techniques

The two major techniques used at the present time for measurement of fluctuating flows are hot wire anemometry and laser Doppler anemometry. The hot wire anemometer relies on the rate of heat loss from an element inserted in the flow whose temperature is deduced from its resistance. In the laser Doppler anemometer the instantaneous velocity is deduced from the Doppler frequency shift of light scattered from particles moving with the fluid. Laser techniques offer one major advantage over the hot wire method in that there is no solid

sensing element in the flow; a double advantage because hot wire probes and their supports may be large enough to disturb the flow and, while the wire itself is small, it is therefore also temperamental. On the other hand, since in laser Doppler anemometry the instantaneous velocity of the flow is deduced from the particles moving with the fluid, and that natural concentration of appropriately sized particles is rarely sufficient for an acceptable data rate, there is usually a need for addition of seeding particles to the flow. The generation of sufficiently small particles in adequate concentration can pose many difficulties and requires special generator designs (Melling and Whitelaw (55), Durst et al. (56), Melling (57)). In the present studies, however, the non-availability and expense of such instrumentation prevented the use of laser techniques.

5.2.1 Physical characteristics of hot wires

Hot wire anemometers are instruments which rely on the rate of heat loss from a probe to determine the instantaneous velocity of the flow in the immediate vicinity of the probe. The sensing element of a hot wire anemometer is a short length of small diameter wire that is heated electrically and is supported by two prongs. An example of a typical probe can be seen in Fig. 5.1.

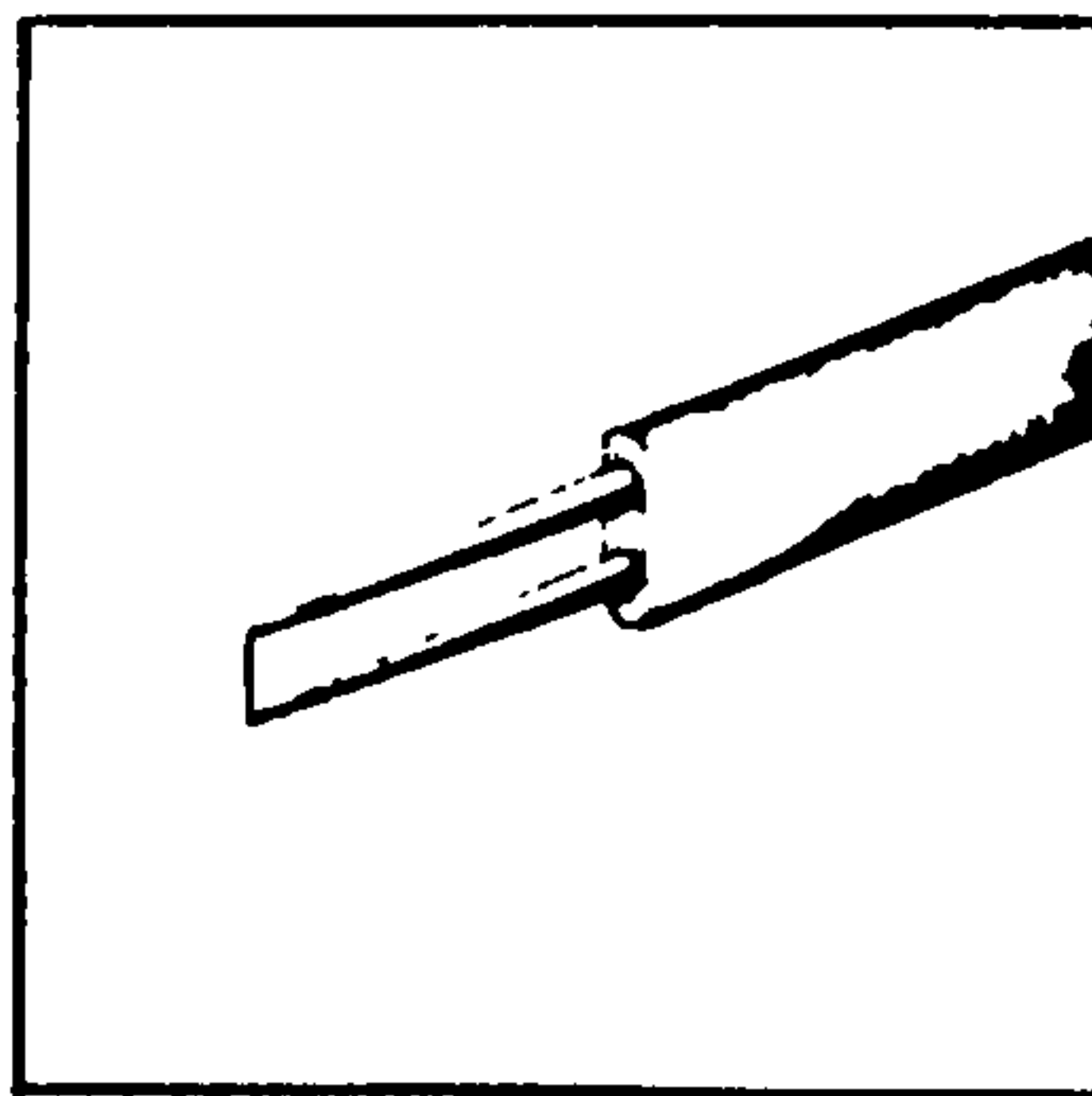


Fig. 5.1 A typical miniature wire probe.

sensing element in the flow; a double advantage because hot wire probes and their supports may be large enough to disturb the flow and, while the wire itself is small, it is therefore also temperamental. On the other hand, since in laser Doppler anemometry the instantaneous velocity of the flow is deduced from the particles moving with the fluid, and that natural concentration of appropriately sized particles is rarely sufficient for an acceptable data rate, there is usually a need for addition of seeding particles to the flow. The generation of sufficiently small particles in adequate concentration can pose many difficulties and requires special generator designs (Melling and Whitelaw (55), Durst et al. (56), Melling (57)). In the present studies, however, the non-availability and expense of such instrumentation prevented the use of laser techniques.

5.2.1 Physical characteristics of hot wires

Hot wire anemometers are instruments which rely on the rate of heat loss from a probe to determine the instantaneous velocity of the flow in the immediate vicinity of the probe. The sensing element of a hot wire anemometer is a short length of small diameter wire that is heated electrically and is supported by two prongs. An example of a typical probe can be seen in Fig. 5.1.

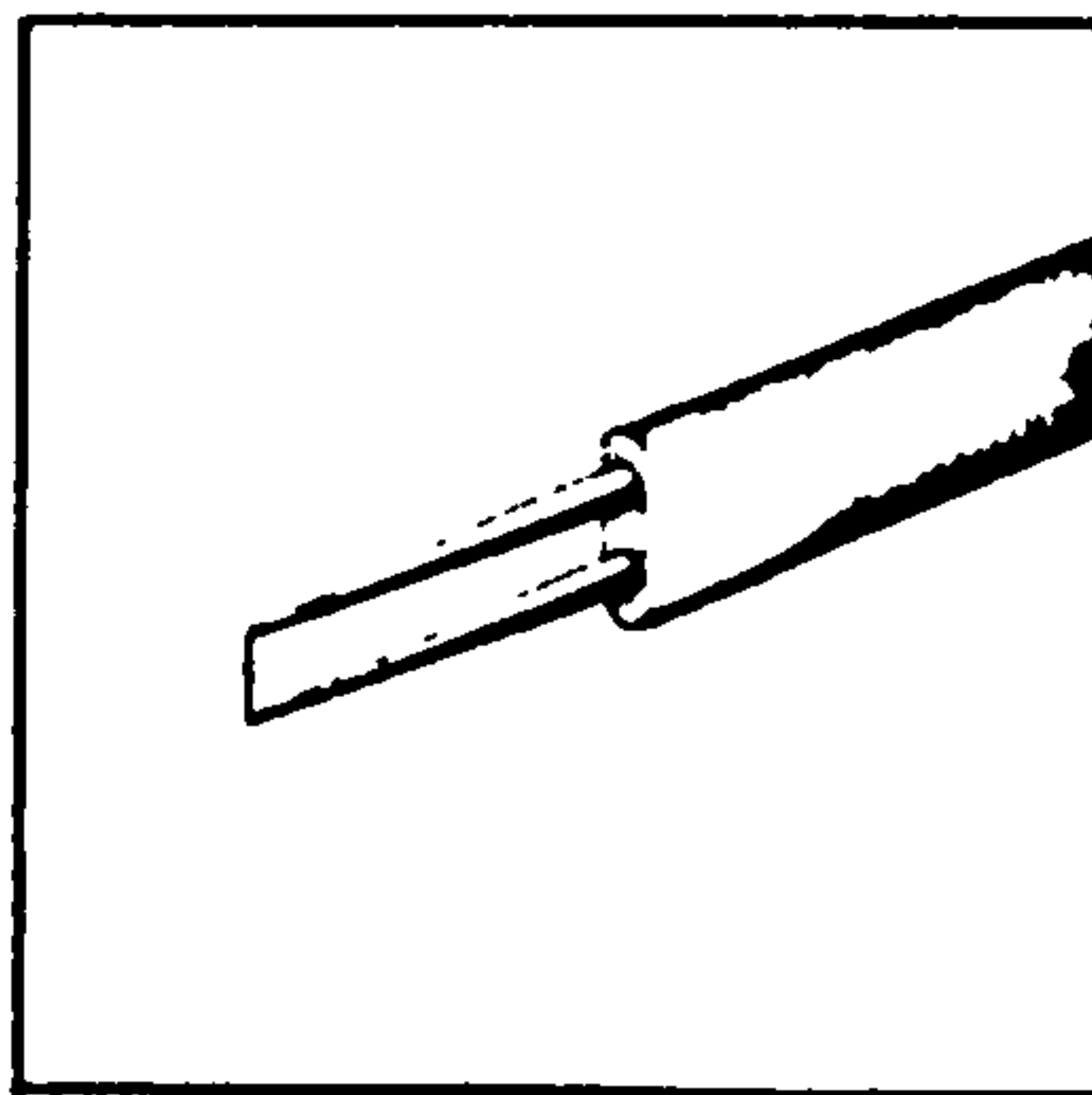


Fig. 5.1 A typical miniature wire probe.

It is usually desirable to have a sensing element that is several hundred times diameter long. This ensures that most of the heat loss is due to the velocity dependent convection and not due to the conductive losses to the end supports. On the other hand, the desire to measure the velocity at a point in space necessitates that the length be small, e.g. $l/d \approx 200$, where l and d are length and diameter of the wire respectively. Thus, very fine wires of diameter $5 \mu m$ or less are usually used.

5.2.2 Principles of hot wire operation

Hot wire anemometry utilizes the convective heat transfer from the heated sensor to the surrounding fluid to measure the velocity. The resistance of the sensor element R is related to its temperature T by

$$R = R_0 \left[1 + \alpha (T - T_0) \right] \quad (5.1)$$

where R_0 and T_0 are the average resistance and temperature at a reference condition, and α is the temperature coefficient of resistivity. Hence, an important parameter governing the operation of hot wire is the overheat ratio defined as

$$a_T = (T - T_0) / T_0 \quad (5.2)$$

where, usually, the ambient temperature is used for the reference condition (i.e. T_0).

The heat transfer mechanisms are based on introduction of heat into the sensing element by Joule heating, and its loss by convection, conduction and radiation.

Usually the sensing element radiates only a very small amount of energy to its surroundings (i.e. much less than 0.1% of the convection losses under normal operating conditions) and hence can be neglected.

In order to maximize the sensor sensitivity to the velocity,

hot wire anemometers normally operate in a forced convection regime. However, at low velocities (i.e. small Reynolds number) free convection can be important. See Collis and Williams (Ref. (58)).

The rate of heat transferred by forced convection from hot wires depends primarily upon the velocity and fluid temperature. The heat loss by convection is given by

$$q_c = h S (T - T_o) \quad (5.3)$$

where h is the convective heat transfer coefficient and S is the surface area through which the heat is transferred. The forced convection losses can be expressed non-dimensionally in terms of the Nusselt number. Therefore, for a cylinder of diameter d ,

$$Nu = h d / K_f \quad (5.4)$$

where K_f is the thermal conductivity of the fluid.

The convective heat losses, and hence the Nusselt number, depend upon almost every possible parameter of the fluid, as well as the properties of the heated element. This can, however, be avoided in the ensuing analysis by making some approximations, and hence reduce the relationship to

$$Nu = Nu (Re, Pr, a_T) \quad (5.5)$$

This is the functional form of the convection heat transfer that is valid for incompressible fluids under restrictions such as neglecting the effects of Grashof number, Knudsen number and aspect ratio of the wire. In addition, the hot wire angle of inclination and the ratio of the specific heats of the fluid is assumed to be constant (Emrich (59)).

Even after the restrictions mentioned, there is still a lack of agreement in the literature concerning the exact form of the relationship expressed in equation 5.5 (Corrison (60), Bourke et al (61),

Collis and Williams (58)). The values of the coefficients and the form of the heat transfer equation become even more diverse when the other variables are included. These variation and deviation from an exact universal equation necessitate that each hot wire be calibrated individually before it can be used to record and interpret data. In addition, the heat loss through the substrate and ends of the wire (i.e. axial conduction along the length of the wire) can be accounted for experimentally by calibration.

5.2.3 Constant temperature anemometer

The simplest and earliest scientific use of hot wire anemometry utilizes a constant current mode of operation (Schubauer (62)). With a suitable choice of voltage and resistance sufficient current can be passed through the wire to cause an appreciable rise in temperature, at which stage the wire resistance, which is a function of temperature, will have a particular value. The voltage across the probe will then be a function of this wire resistance. Therefore the voltage variation will become a function of flow velocity which can be found by calibration.

There are, however, two fundamental disadvantages to this system. Firstly, the limited flow range of the device due to the rapid fall of device sensitivity with velocity, and secondly the limited frequency response of the system imposed by the thermal inertia of the probe.

The two fundamental disadvantages of the constant current anemometer are overcome in the present study by using a system which maintains the probe at a constant temperature in excess of the fluid temperature.

The basic elements of a constant temperature anemometer circuit are a differential dc amplifier, the sensing element and reference voltage. A typical circuit is shown in Fig. 5.2, where the sensor is placed in one leg of a wheatstone bridge. The current through the element gives a voltage e_1 . A reference voltage is used by the feedback amplifier to determine how much the resistance, and hence temperature, of the sensor has changed. This can conveniently be obtained as the bridge voltage e_2 . These two voltages form the inputs to the dif-

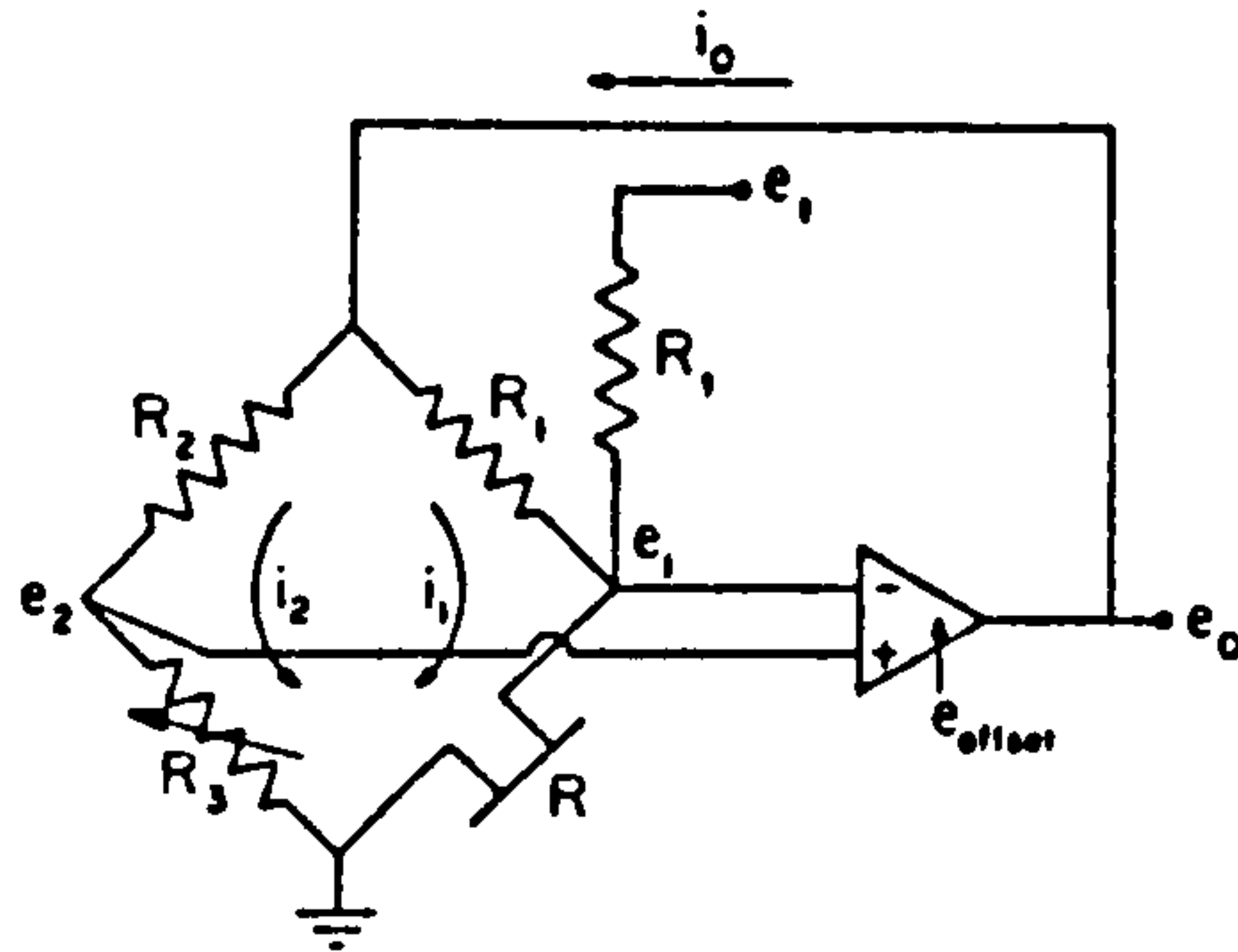


Fig. 5.2 Schematic of a constant temperature anemometer circuit with provision for a test signal e_1 .

ferential amplifier. The amplifier's fluctuating output current i_o is inversely proportional to the resistance change in the sensor. This current is fed back into the top of the bridge as shown, and restores the sensor's resistance and temperature to their original values. Since the response of the amplifier is quite fast, its fluctuating current maintains the sensor at a constant temperature. The output voltage e_o can then be related to fluid velocity by calibration. Hot wire sensors typically have a low resistance of $10 - 100 \Omega$ and require $5 - 50$ mA from the amplifier to generate moderate overheat values. To efficiently use the available current from the amplifier, the resistance in the left leg of the bridge in Fig. 5.1 is usually larger than that in the right leg. R_1 / R_2 is called the bridge ratio and typically has a value of 10.

The hot wire bridge circuit is balanced whenever

$$R R_2 = R_1 R_3 \quad (5.6)$$

To accommodate different wire resistances and/or to set overheat ratio at least one leg has a variable resistor as shown in Fig. 5.2.

The advantage of this system is that the probe temperature remains virtually constant, hence overcoming some of the limitations on frequency response arising from the thermal inertia of the probe.

The higher the gain of the amplifier, the smaller will be the error voltage required to produce a given output voltage, and hence the faster will be the system response, which is ideal for the high frequency response needed for measurement of fluctuating flows. Moreover, since the probe temperature remains constant, the problems of limited flow range normally encountered with constant current systems are not experienced.

5.3 Equipment and set up

5.3.1 Instrumentation for hot wire velocity measurements

The equipment used for the measurement of point velocities were based on the DISA 55M system. The 55M system employs the building block principle in which a complete measuring system is composed of a number of individual units. This makes it possible to set up for a particular measurement programme a combination of instruments that will present an optimum, not only of quality but of operating convenience as well. The arrangement of the units making up the present system can be seen in Fig. 5.3.

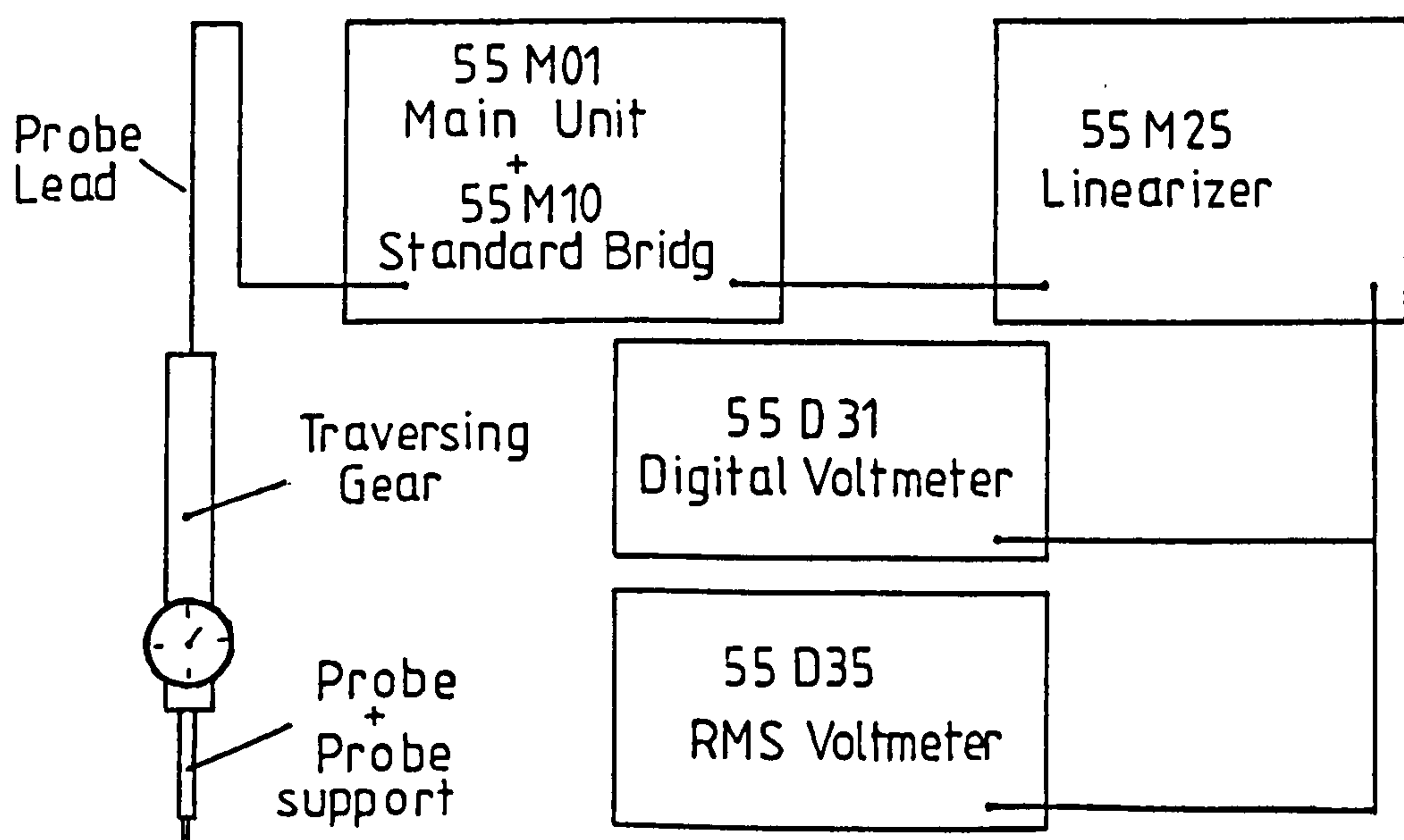


Fig. 5.3 Arrangement of the equipment for point measurement of velocity and its fluctuation.

Probe and traversing gear

The probe used was of the single element general purpose type (DISA 55P11) (see Fig. 5.1), consisting of a 5- μ m-dia, 1.25 mm long platinum plated tungsten wire, welded on a 5 mm long prongs and mounted on 4 mm-dia probe supports.

The probe support was incorporated into the traversing gear by a 215 mm long mounting tube carrying the probe lead. A specially modified vernier was used as a traversing gear, allowing for traversing in 0.05 mm increments (Fig. 5.3).

Main unit and standard bridge

The 55M01 main unit and the 55M10 standard bridge contained all circuits required for operating an anemometer: amplifier, filter, decade resistance, square-wave generator, protection circuits and other auxiliary circuits. The hot wire probe was connected to the main unit by a 5 m long lead, and the output voltage from this unit was fed to the linearizer.

Linearizer

The output voltage from a constant temperature anemometer is a non-linear function of the flow velocity. This non-linearity has little influence on measurements at low levels of turbulence, where a small portion of the calibration curve may be considered sufficiently linear. At higher levels of turbulence, however, such as are encountered in some regions in the present study, distortion caused by the non-linearity of the calibration curve can become significant. In addition, measurements at varying mean flow velocities are made difficult by sensitivity variations. In many cases mathematical post-processing of such outputs may be unsatisfactory, and automatic electronic linearization during measurement is often preferable.

The 55M25 linearizer (Fig. 5.3) accomplishes the linearization of the anemometer signal by means of an analogue computer having a transfer function composed of an exponential and a square root function having the following form:

$$V_{LIN} = K_1 (V_{CTA} + V_o)^m (V_{CTA} - V_o)^m + K_2 (V_{CTA} - V_o)^{1/2} \quad (5.1)$$

where K_1 and K_2 are constants depending on the measuring conditions. V_{CTA} and V_0 are anemometer output voltage and output voltage at zero flow and m is the exponent obtained from the Kings law heat transfer expression.

Digital voltmeter and RMS voltmeter

55D31 digital voltmeter and 55D35 RMS voltmeter was used to obtain the time-averaged and RMS value of the linearized anemometer output signal respectively, and hence the mean velocity and corresponding turbulence content of the flow under investigation.

5.3.2 Testing set up

Perspex models

The Perspex models used for velocity measurements were identical to the ones used in flow visualization tests. Their only additional feature was the introduction of 4 mm holes at appropriate stations to act as access ports for the hot wire probe, as illustrated in Fig. 5.4

I_1 and I_2 provide access for traversing of the probe through the inlet plane, whereas positions 1 to 9 upstream of the cylinder were made accessible through the six holes shown in Fig. 5.4. Flow measurement around the cylinder head was made possible through holes A to D.

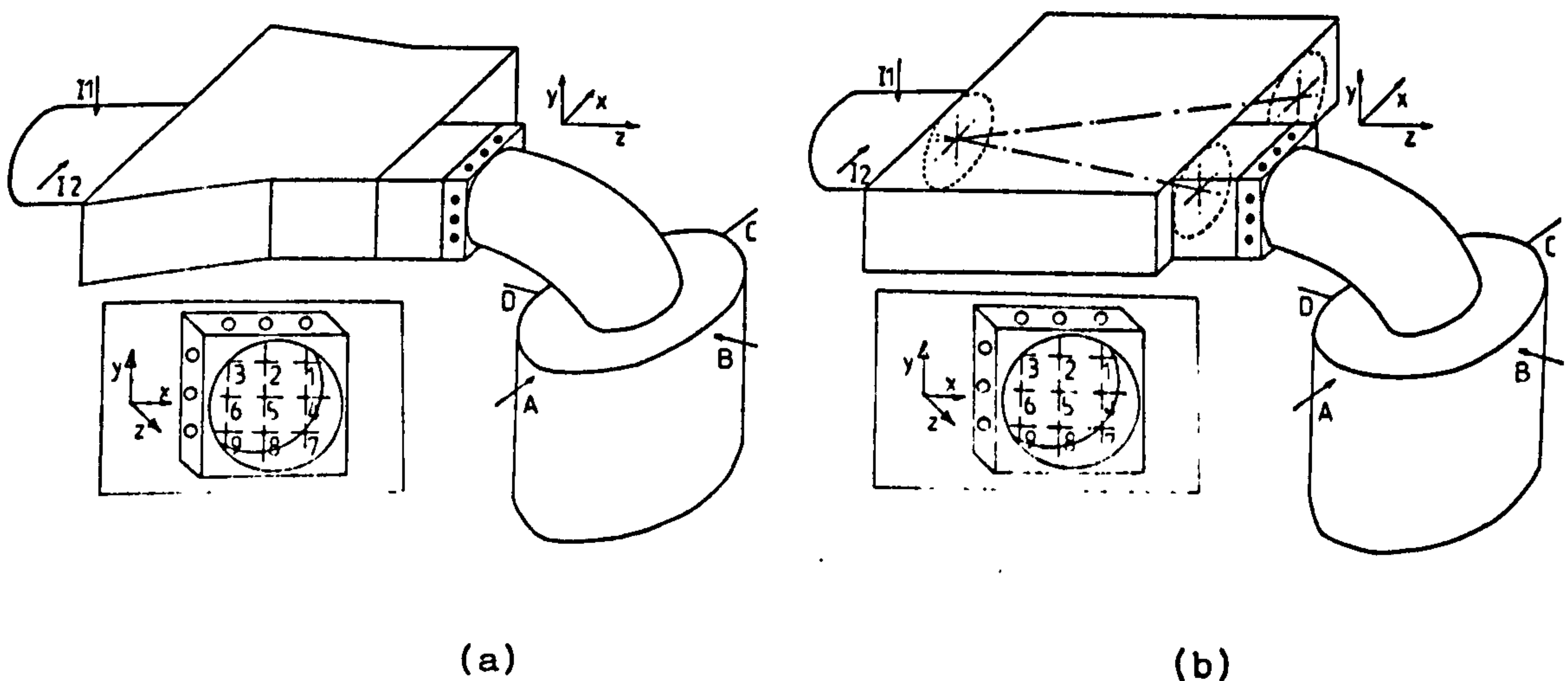


Fig. 5.4 Traversing stations in Perspex models for hot wire velocity measurements.

Air flow rig

The air flow through the models was produced by a T6BVC blower/exhauster acting as a vacuum pump. The rate of air flow through the system was controlled by a ball valve, and pressure variations across an orifice plate upstream of the valve were used to measure the rate of flow of air through the rig (Fig. 5.5).

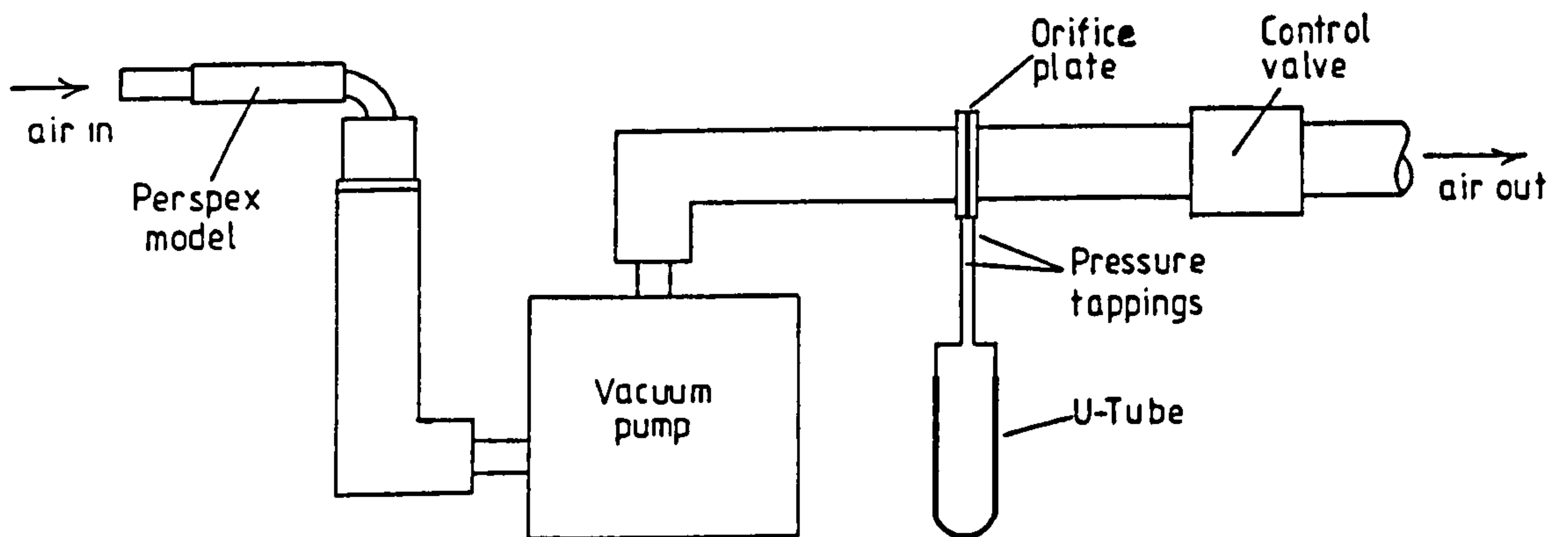


Fig. 5.5 Schematic of the air flow test rig.

5.4 Experimental procedure

5.4.1 Hot wire calibration

As it was mentioned in section 5.2.2 the hot wire anemometer system has to be experimentally calibrated before proceeding to the actual flow measurements in the models.

The overheat ratio of the system was set to 0.625 by setting the operating temperature of the sensor to 473 K and using the ambient temperature for the reference conditions. The probe resistance for this setting was found to be 6.86Ω , using the following relationship:

$$R = R_1 + R_0 (1 + \alpha(T - T_0)) \quad (5.8)$$

where

R = sensor resistance at operating temperature of 200°C
 R_0 = sensor resistance at reference ambient condition $3.8\ \Omega$
 R_1 = resistance of 5 m probe lead $0.6\ \Omega$
 α = temperature coefficient of resistance $0.36\ \% / ^{\circ}\text{C}$
 T = operating temperature of the sensor 200°C
 T_0 = reference ambient temperature 20°C

In order for the system to provide a high frequency response, the gain of the amplifier needs to be high. At these high amplification gains the system has to be checked and calibrated for stability.

The dynamic calibration of the system was obtained by using an electronic test signal, namely a square-wave generated by a square-wave generator incorporated in the 55 M 01 main unit.

The probe was exposed to a constant velocity of 50 m/s (i.e. maximum velocities encountered during the measurement) using the 55D45 calibration nozzle unit. The frequency of the signal from the square-wave generator was set to 3 KHz, and an oscilloscope was used to monitor the output voltage. Oscillations at high gain settings were compensated by adjustment of a pair of cable compensators and the best possible frequency response for the system was found to be about 80 KHz.

For the static calibration of the probe the DISA 55D45 nozzle unit was used to produce a variable velocity, low turbulence level air jet, in which the hot wire was placed at right angle relative to the axis of the jet. Given the reference atmospheric pressure and temperature, the jet velocity during calibration was determined from the pressure drop across the nozzle. A computer generated table of nozzle pressures against jet velocities for different atmospheric conditions was employed to produce jet velocities ranging between 2 to 50 m/s.

The non-linear signal from the anemometer was linearized using the DISA 55M25 linearizer and the calibration chart for the hot wire probe can be seen in Fig. 5.6

During the course of the experiments it was found necessary to adjust the linearizer from time to time to compensate for the day to day changes in atmospheric conditions which affected the calibration.

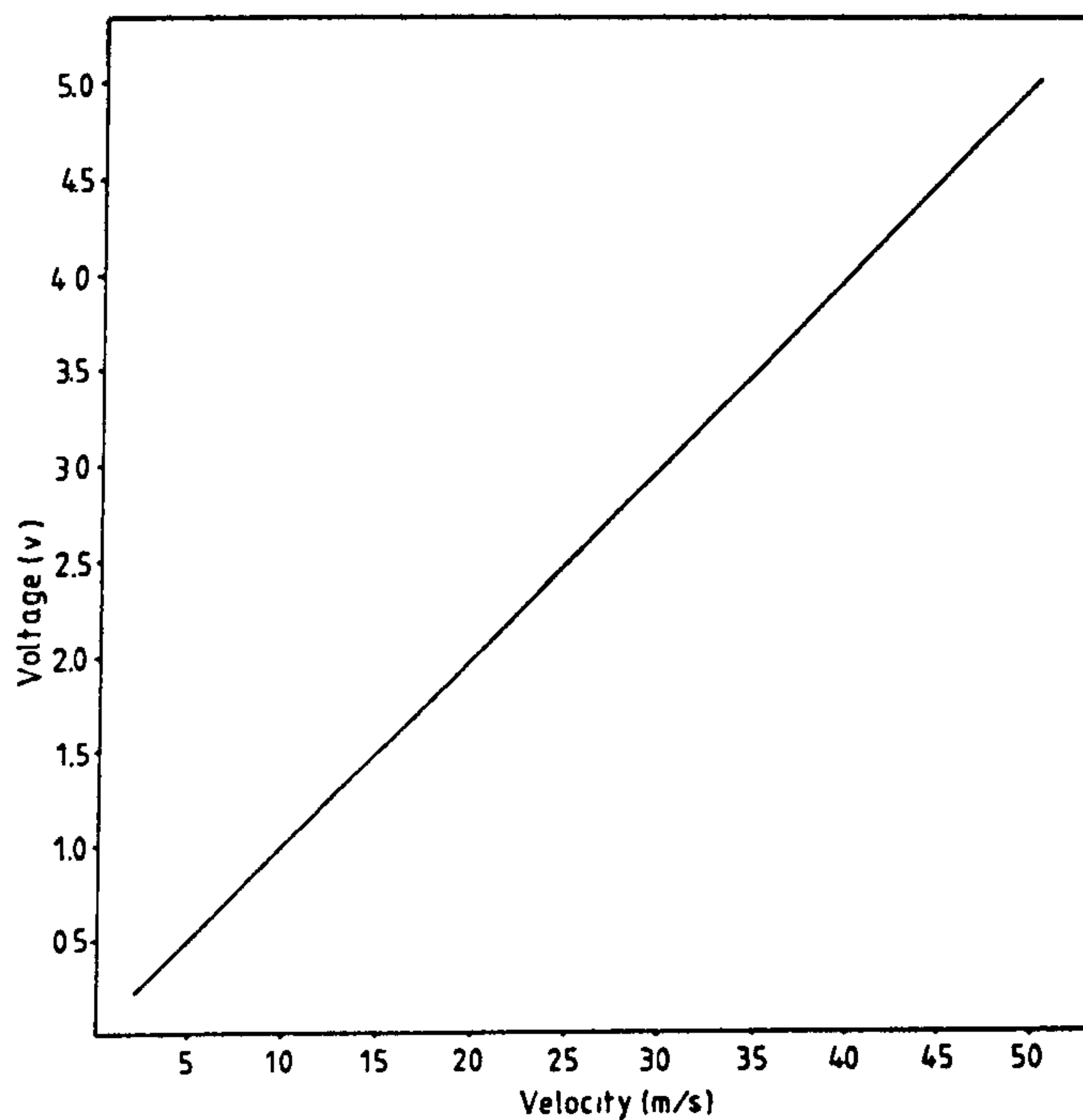


Fig. 5.6 Linearized calibration chart for the hot wire probe.

5.4.2 Steady flow tests

A schematic arrangement of the steady flow rig can be seen in Fig. 5.7. Perspex models of the throttling systems and cylinder were located in the suction pipe of the rotary exhaust/blower, using a 5 cm pipe connected to the cylinder by an adaptor. The flow rate through the system was varied by adjusting the control valve downstream of the orifice plate.

The pressures P_1 and P_2 , at locations 1 and 2 in the models, were measured using a standard electronic manometer.

Nine throttle settings, based on effective throat areas and the corresponding pressure drops (i.e. $P_1 - P_2$), were tested. These settings are identified in Table 5.1. The four flow rates used in the tests were broadly based on average flow rates encountered during the induction stroke at various engine speeds. These will be discussed in more detail later.

Eight of the throttle settings tested were identical to the ones employed in the flow visualization tests and were based on the similarity of the effective throat areas. In addition, the similarity

of pressure drop across the two alternative systems was also considered (i.e. comparable $P_1 - P_2$), leading to an additional test on the butterfly valve system at $A_t/A_i = 0.74$ (cf. Table 5.1).

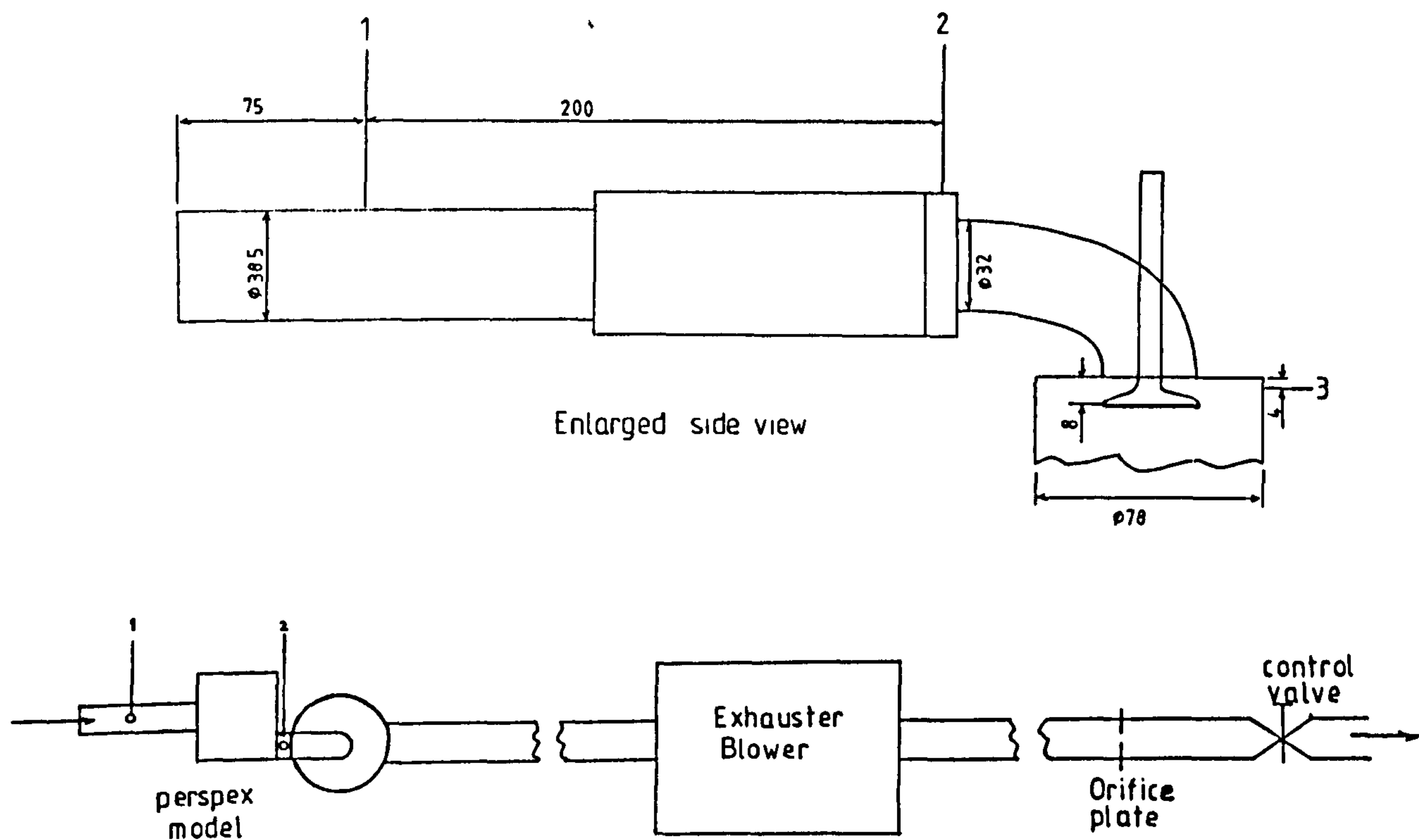


Fig. 5.7 Schematic arrangement of the steady flow rig. (All dimensions in mm).

A_t/A_i	Butterfly valve system		Variable ramp system		Air flow rate Q l/s
	β	ΔP (mm H_2O)	Θ	ΔP (mm H_2O)	
0.13	30	527	110	527	11.6
0.36	50	124	130	124	17.5
0.58	65	75	150	41	23.2
0.74	75	41	---	---	23.2
WOT	90	54	180	44	29

Table 5.1 Throttle settings and flow rates tested. β = butterfly valve angle; Θ = ramp angle; $\Delta P = (P_1 - P_2)$ the pressure drop across the throttling system; A_t = effective throat area; A_i = inlet pipe area.

For each of the nine throttle settings the hot wire probe was traversed through planes 1, 2 and 3 (Fig. 5.7) and measurements were made at the locations indicated in Fig. 5.8. The result of these measurements can be seen in Fig. 5.9 to 5.47 at the end of this chapter.

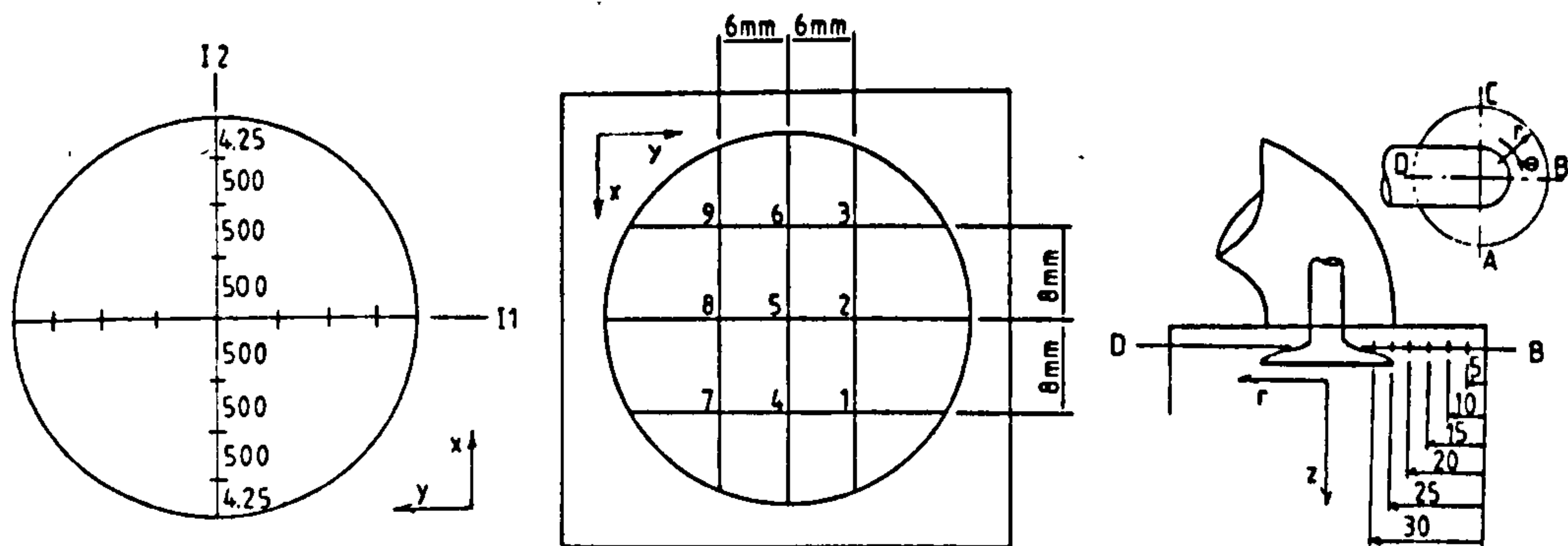


Fig. 5.8 Locations of the hot wire measurements at planes 1, 2 and 3. (All dimensions in mm).

5.4.3 Choice of flow rates tested

In a spark ignition engine it is possible to distinguish four different contributions to the overall pressure drop as air flows from the atmosphere to the engine cylinder through the induction system:

- 1) engine intake, air filter and carburettor or throttler,
- 2) friction in manifold pipes and in ports,
- 3) manifold junctions, and
- 4) inlet valve.

It is these restrictions in the system that determine the volumetric efficiency and hence the rate of flow through the system at different loads and engine speeds.

Under steady flow conditions we may take a control volume around the complete throttling system and select the inflow and outflow planes arbitrarily, but in such a manner that the flow may be assumed

to be one-dimensional in these planes. It may then be assumed that the pressure drop ΔP across the throttle is of the form:

$$\Delta P = f_c \frac{1}{2} \rho u^2 \quad (5.9)$$

where ρ and u are the density and velocity upstream of the throttle and f_c is a constant or pressure-loss coefficient. This pressure-loss coefficient will be dependent on the Reynolds number, the Mach number at the throttling device and the throttle position

$$f_c = \phi (\text{Mach number, throttle position, Reynolds number}) \quad (5.10)$$

Under unsteady flow conditions the pressure wave action at a throttling system is similar to wave action at a gauze (Benson and Barnah (63), Benson et al (64)). A pressure wave incident on the throttling system will be reflected with increasing amplitude and transmitted through the throttle with decreasing amplitude. The increase in amplitude of the reflected wave is typically greater than the decrease in amplitude of the transmitted wave. There is therefore a build-up of pressure upstream of the throttle due to such non-steady flow. This would mean that the steady flow resistance coefficient may not then be equal to the dynamic resistance coefficient.

In addition to the factors mentioned above the flow rate in a firing engine depends on load, torque, engine capacity and speed. In consequence, no generally appreciable, meaningful correlation can be made between engine speed, throttle setting and flow through an engine, without extensive experimental data.

In the absence of such experimental data it was decided, for purposes of simulation, to base the average flow rates during the induction stroke of the engine on engine speed and corresponding mean piston velocities. The simulated engine was assumed to be a four-cylinder, 1400 cc engine, with bore-to-stroke ratio of 1.08 (i.e. bore diameter 78 mm, stroke length 72 mm). The effects of restrictions on volumetric efficiency were neglected, given the difficulties outlined earlier. Accordingly, the average flow rate Q during the induction stroke becomes:

$$Q = A_{\text{pis}} \bar{V}_{\text{pis}} \quad (5.11)$$

where A_{pis} is the piston area, based on bore diameter, and $\bar{V}_{\text{pis}}$ is the average piston velocity, based on engine speed $N(\text{r.p.m.})$ and stroke length $S(\text{m})$.

$$\bar{V}_{\text{pis}} = 2 \frac{N S}{60} \quad (5.12)$$

The flow rates used in the tests (Table 5.1) were based on engine speeds of 1000, 1500, 2000 and 2500 r.p.m., resulting in flow rates of 11.6, 17.5, 23.2 and 29 l/s.

The correlation between flow rate and throttle setting was based on the use of higher engine speeds with lower* throttle setting which is more representative of a firing engine.

5.4.4 Hot wire orientation and interpretation of its measurements

A cylindrical hot wire is entirely insensitive to changes of flow direction in the plane normal to the axis of the wire. In addition, according to the equations of motion, the flow over an infinitely long wire in the plane normal to its axis should be independent of the velocity along the axis of the wire. The heat transfer, on the other hand, depends upon both the axial and normal components of the flow.

The resolution of mean velocity and turbulence intensity is dependent on the mean velocity being of known direction and much larger in magnitude than the size of turbulent fluctuations. These two conditions are seldom satisfied in the cylinder of the Perspex models tested. Indeed, it is conceivable that at certain locations there is no mean flow, even when the turbulent velocities may be quite large. In such an instance the hot wires's insensitivity to flow direction will cause the anemometer to indicate a false mean velocity with a corresponding reduction in the true turbulence intensity.

The three most common configurations for hot wire probes are:

* larger throttle opening

1) A single wire normal to the stream, used in the present studies.

2) A pair of wires arranged in a "X" at approximately 45° to the flow direction. Ideally, one wire of an "X" probe in the XY plane responds to $u+v$ and the other to $u-v$, according to the simple cosine law, so that the difference between the two wire signals is $2v$. In practice, however, the wires can not be made electrically and aerodynamically identical, and the additional instrumentation required can be preventative in the use of such configurations.

3) The slant wire probe, which looks like half an "X" probe and responds roughly to $u+v$, and if rotated 180° about the probe axis will respond to $u-v$. When used in this way, the slant wire probe simulates an "X" probe with two perfectly matching wires and saves on the need for additional instrumentations. But, of course, it can not be used for instantaneous measurement of the v components.

Despite the fact that crossed wires and slant wire probes provide a certain sensitivity to flow direction, the lack of sufficient instrumentation and geometric accessibility of the flow resulted in the adaptation of a standard single wire in the present studies.

Because of these limitations, it was necessary to interpret the velocity and turbulence intensities determined from the anemometer output at certain locations (such as plane 3 inside the cylinder) as being merely "representative" measures of the hot wire response to its environment. This is not to say that such measurements are meaningless, because they are truly representative of the "effective" mass flow conditions sensed by the probe, such that in most instances they give at least an indication of the flow structure in the cylinder.

Assuming that there is a mean velocity of large magnitude relative to the turbulent fluctuation in a given plane normal to the axis of the hot wire, and supposing for a moment that the instantaneous component of velocity along the axis of the wire has little or no effect on the wire response, then the wire will respond to

$$\sqrt{(U + u)^2 + v^2} \quad (5.13)$$

where U is the mean velocity, u the fluctuating component in the

direction of mean velocity, and v the fluctuating component normal to it. If the ratio of the fluctuating component to mean velocity is small, equation 5.13 can be expanded to

$$U \left[1 + \frac{u}{U} + \frac{1}{2} \frac{v^2}{U^2} \right] \quad (5.14)$$

neglecting third and higher orders. Taking a time mean value for equation 5.14 we find

$$U \left[1 + \frac{1}{2} \frac{\overline{v^2}}{U^2} \right] \quad (5.15)$$

Therefore, if the mean velocity U is being measured, the hot wire result is larger by the factor

$$\left(1 + \frac{1}{2} \frac{\overline{v^2}}{U^2} \right) \quad (5.16)$$

Subtracting equation 5.15 from 5.14 we find that the apparent fluctuating part of the signal is given by

$$u + \frac{1}{2} (v^2 - \overline{v^2}) / U \quad (5.17)$$

Similarly, when u^2 is being measured, the hot wire response is too large by the factor

$$1 + \frac{\overline{uv^2}}{U\overline{u^2}} + \frac{\overline{v^4} - (\overline{v^2})^2}{4U^2 \overline{u^2}} \quad (5.18)$$

and if $\sqrt{u^2}$ is being measured, the result is modified by the factor

$$1 + \frac{1}{2} \frac{\overline{uv^2}}{U\overline{u^2}} + \frac{1}{8} \left[\frac{\overline{v^4} - (\overline{v^2})^2}{4U^2 \overline{u^2}} \right]^2 - \frac{1}{8} \left[\frac{\overline{uv^2}}{U\overline{u^2}} \right]^2 \quad (5.19)$$

From equation 5.16 and 5.19 it can be seen that at low relative turbulent intensities (i.e. low $\sqrt{\overline{u^2}}/U$ and $\sqrt{\overline{v^2}}/U$), the hot wire signal can provide results close to the true mean velocity and its fluctuating part. In practice, however, there is some uncertainty about the quantitative value of the turbulence results in regions where turbulent intensities are larger than 15 to 20% or so. This is especially true when measuring the fluctuating component (i.e. $\sqrt{\overline{u^2}}$).

The interpretation of the hot wire signals in the present study depended to a large extent on the particular region in which the flow was measured, the orientation of the hot wire relative to the flow and the throttle setting.

In plane 3 (Fig. 5.8), measurements of mean velocity and mean square turbulence fluctuation were made in two planes normal to the axis of the wire, namely planes (r, θ) and (r, z) . These measurements can not be interpreted unambiguously as values of mean velocity and fluctuations in the planes mentioned. This is mainly due to the high turbulence intensities and the absence of large mean velocities with known direction in this region. The measurements, on the other hand, can be interpreted as representative of the overall mass flux conditions sensed by the probe. The results of these measurements can be seen in Figs. 5.29 to 5.46.

In plane 1 (Fig. 5.8), in contrast to plane 3, the flow is predominantly in the z -direction, with small turbulence intensity values. The measurements were therefore made with the axis of wire normal to the z -axis of flow. The result of these measurements can be seen in Figs. 5.9 and 5.10.

In plane 2 (Fig. 5.7), the nature of flow depends to a large extent on the throttle setting and the type of device. Therefore, measurements had to be interpreted differently for different test settings. At this plane, measurements were made in three planes normal to the axis of the wire, namely planes (x, y) , (x, z) and (y, z) . As the flow for all of the nine settings tested was predominantly in the z -direction, the effect of heat transfer along the axis of the wire was assumed to be negligible in planes (x, z) and (y, z) . In plane (x, y) , however, the heat transfer effect on the hot wire signal can not be assumed negligible. This is due to the large component of

velocity along the axis of the wire (i.e. z-direction). In addition, the lack of any significant mean velocity in this plane adds to the ambiguities of the measured signal. The measurements in the (x,y) plane are therefore presented as a matter of record and will not be discussed in any detail in the following section, or for that matter compared with the computer results in Chapter 6. The results of measurements in plane 2 can be seen in Figs. 5.11 to 5.28.

5.5 Discussion of results

The profiles of axial velocities W and the corresponding relative turbulence intensities $\sqrt{w^2} / W$ for the inlet pipe (i.e. plane 1) are shown in Figs. 5.9 and 5.10. The velocity profiles indicate an even distribution of velocity components for all the flow conditions tested. The maximum variation between the center line velocity component and the near-wall measured values was found to be around 15%. The measured mean velocities of 10, 15, 20 and 25 m/s correspond well with the volumetric flow rates of 11.6, 17.5, 23.2 and 29.0 l/s respectively. The measured turbulence fluctuation components (r.m.s.) for center line and near-wall traversing positions were of the order of 0.05 and 1.5% of the local mean velocities respectively. Overall, the hot wire results indicate that the distribution of axial velocities and turbulence intensities are to a large extent independent of the type of device or throttle setting in this region.

In contrast to plane 1, the profiles of mean velocities and turbulence intensities shown in Figs. 5.11 to 5.28 indicate a great dependency on the device type and throttle setting. As can be seen in Fig. 5.11, for the ramp device at 110° the velocity measurements in (z,y) and (z,x) planes vary dramatically from one position to another, with high velocities of around 35 to 50 m/s at plane (1,4,7), decreasing sharply to around 15 to 20 m/s at plane (2,5,8), and then apparently rising again at plane (3,6,9), but more plausibly indicating a region of reverse flow at plane (3,6,9). These pronounced velocity gradients are the result of the strong recirculation zone created by the high velocities of about 100 m/s over the apex of the ramp, as observed in the photographs of the flow visualization studies.

The recirculation is predominantly in the (z,x) plane, with relatively little cross flow in (z,y) plane, hence the larger velocity components recorded in the plane of recirculation. The turbulent intensity profiles (Fig. 5.12) suggest high levels of mixing, with fluctuating components of velocity varying between 20 to 50% of the local mean velocities.

The velocity profiles for the butterfly valve at 30° (Fig. 5.19) show a much more uniform distribution of flow in comparison to the variable ramp device. The velocity profiles show little or no difference between the measured velocity components in (z,y) or (z,x) plane. This, to a large extent, indicates the non-directional nature of flow in (x,y) plane. The turbulence intensities (Fig. 5.20) are overall much smaller in value than those for the variable ramp device, these reductions being of the order of 15 to 50% depending on the probe position. This also corresponds well with the qualitative pictures of the flow obtained in the previous chapter.

The recorded pressure drops across both throttling systems (i.e. $P_1 - P_2$) (Table 5.1) were found to be identical. This indicated to a large extent the similarity of resistance coefficient for both systems at these high throttle settings.

Fig. 5.13 shows the velocity profiles for the variable ramp device at ramp angle 130°. These profiles are to a large extent similar to the ones for ramp angle 110°, with the exception of no apparent rise in velocities at plane (3,6,9). This indicates a smaller recirculation zone downstream of the ramp, with no indication of reverse flow at plane (3,6,9). The corresponding turbulence intensity values are overall lower for this, the lower throttle setting with their values ranging between 0.1 to 0.45.

The velocity profiles for butterfly valve at valve angle 50° (Fig. 5.21) show a much more even distribution of velocities in comparison with the corresponding ramp setting (i.e. 130°). This is accompanied by lower relative turbulence intensities of around 0.2 (Fig. 5.22). The pressure drops across both devices, however, were found to be very similar (Table 5.1).

The velocity gradients for the variable ramp at an angle of 150° (Fig. 5.15) appear to be less severe, with maximum velocity variations between 25 to 35 m/s. This could be the result of larger effective throat area, and hence a smaller region of flow separation

downstream of the ramp. This is also reflected in the lower turbulence intensity values of 0.05 and 0.25 shown in Fig. 5.16. The velocity profiles for the butterfly valve at valve angle 65° (Fig. 5.23) indicate an evenly distributed flow of about 30 m/s across plane 2, with corresponding turbulence intensities of between 0.1 to 0.2 shown in Fig. 5.24. Overall this indicates a similar turbulence level for both systems at these lower throttle settings.

The pressure losses, however, differed significantly, with pressure drops across the butterfly valve model being about 65% higher than that for the variable ramp model (Table 5.1). This indicates that, whereas at high throttle settings the effective throat area of the device was the predominant factor in the determination of the pressure losses at low throttle settings the overall geometry of the manifold becomes the more predominant factor.

To achieve an identical pressure drop the measurements were made at identical flow rates, but with the butterfly valve set at valve angle 75° . This in effect meant an increase in effective throat area of about 27%. As can be seen in Figs. 5.25 and 5.26, there appears to be little change in velocity profiles, with a slight reduction in the turbulence intensity values in some regions.

At wide open throttle the velocity profiles for the butterfly valve device (Fig. 5.27) show a higher level of variation than that for the variable ramp system (Fig. 5.17). This might be the result of some flow separation at the T-junction upstream. This flow separation was also observed in the flow visualization tests in the previous chapter. The turbulence levels appear to be identical for both systems, with average values for turbulence intensity of about 0.05 (Figs. 5.18 and 5.27).

The pressure drop across the butterfly valve system was recorded to be about 22% higher than that for the variable ramp system (Table 5.1). Taking the previous results also into consideration, this indicates a better volumetric efficiency for the variable ramp device at low and wide open throttle settings.

Unlike planes 1 and 2, where the mean axial velocity and turbulence intensities are relatively well defined, the nature of flow in the model cylinder (i.e. plane 3) is highly anisotropic, which makes the quantitative study of the flow with hot wire single point

measurement very difficult, if not impossible. A general and qualitative interpretation of the measured quantities, however, is attempted here.

The profiles of mean velocity and turbulence intensity are presented for the case of variable ramp geometry in Figs. 5.29 to 5.36, and for the case of the butterfly valve in Figs. 5.37 to 5.46. The basic flow pattern is of a high velocity jet drawn into the cylinder indicated by high velocities near the inlet valve. There are also indications of a strong vortex flow in the corner between the wall and the cylinder head. This is signified by the rise in velocity near the cylinder wall, indicating a strong reverse flow in this region. The main distinguishing feature between the two alternative devices appears in the velocity profile at high throttle settings. As can be seen in Fig. 5.29, there appears to be a strong circumferential variation in axial velocity at a distance of 25 mm from the cylinder wall. To a large extent this must be the result of high velocity gradients upstream of the inlet valve. This is also reflected in the velocity profiles in the (r, Θ) plane of measurement (Fig. 5.30), with high velocity variations at a distance of 30 mm from the wall, which corresponds to a position near the valve stem. In contrast, the velocity profiles for the butterfly valve at 30° (Fig. 5.37 and 5.38) appear to be more evenly matched around the cylinder.

The turbulence intensity levels appear to be evenly matched for both systems, with noticeable reductions in some regions at lower throttle settings. These values, however, are affected strongly by the ambiguity of the hot wire, and hence can not be interpreted as showing similar levels of mixing for the systems.

5.6 Concluding remarks

In view of the inherent limitations in using the hot wire anemometer in the highly anisotropic regions of flow, and the limited amount of data acquired, it is probably unwise to use such measurements to compare the two alternative devices quantitatively. These measurements, however, were primarily meant to complement the flow visualisation results and provide an additional means of validation for the computer

predictions. Accordingly, we restrict comment to generalized statements of the trends observed.

- The high velocities over the apex of the ramp at high throttle settings result in a strong recirculation zone downstream of the ramp. This is indicated by high velocity gradients encountered in the induction manifold and near the inlet valve.

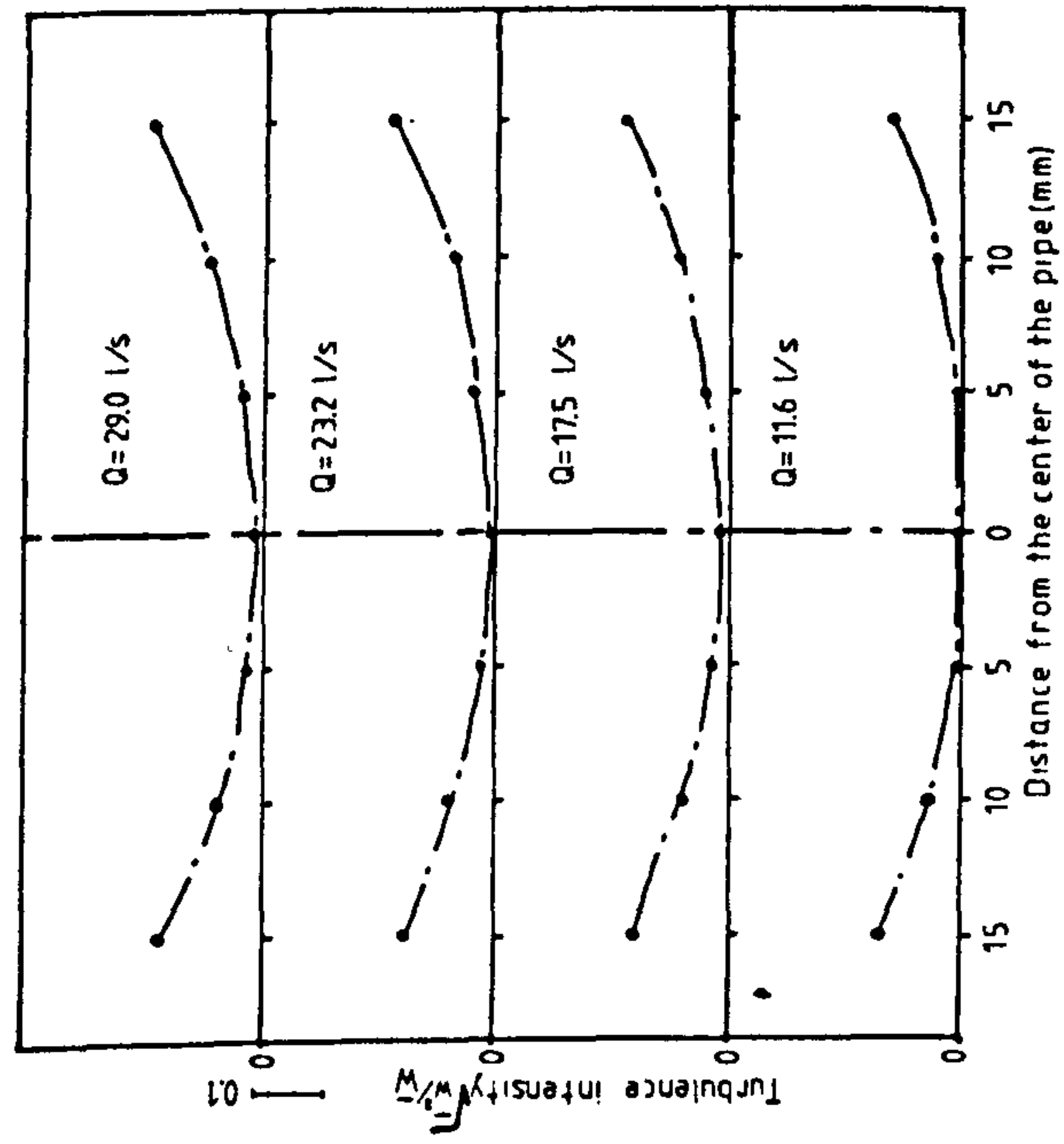
- The velocity profiles over the complete range of butterfly valve settings tested indicated a well defined pipe flow in the induction manifold, with little or no effect carried downstream, from the disturbed velocity profiles at the butterfly valve.

- At high throttle settings, the measured r.m.s. velocity components indicated higher turbulence levels for the variable ramp device in comparison to the more conventional butterfly valve system. This appeared to be mainly the result of stronger recirculating and mixing flows downstream of the ramp.

- At low and wide open throttle settings the turbulence levels and mean velocity profiles in the induction manifolds and near the inlet valves appear to be very closely matched for both throttling devices.

- At high throttle settings the pressure drops across both systems appear to be a strong function of effective throat area. Whereas at low and wide open throttle settings the overall geometry of the devices appeared to become the dominant factor in determining the overall pressure losses, the variable ramp device indicated somewhat better volumetric efficiency.

Overall there was good agreement between the hot wire and flow visualization results, and in that sense the measurements provided a valuable additional means by which to verify the computer predicted results in the following chapter.



Q = Volumetric flow rate through the system.

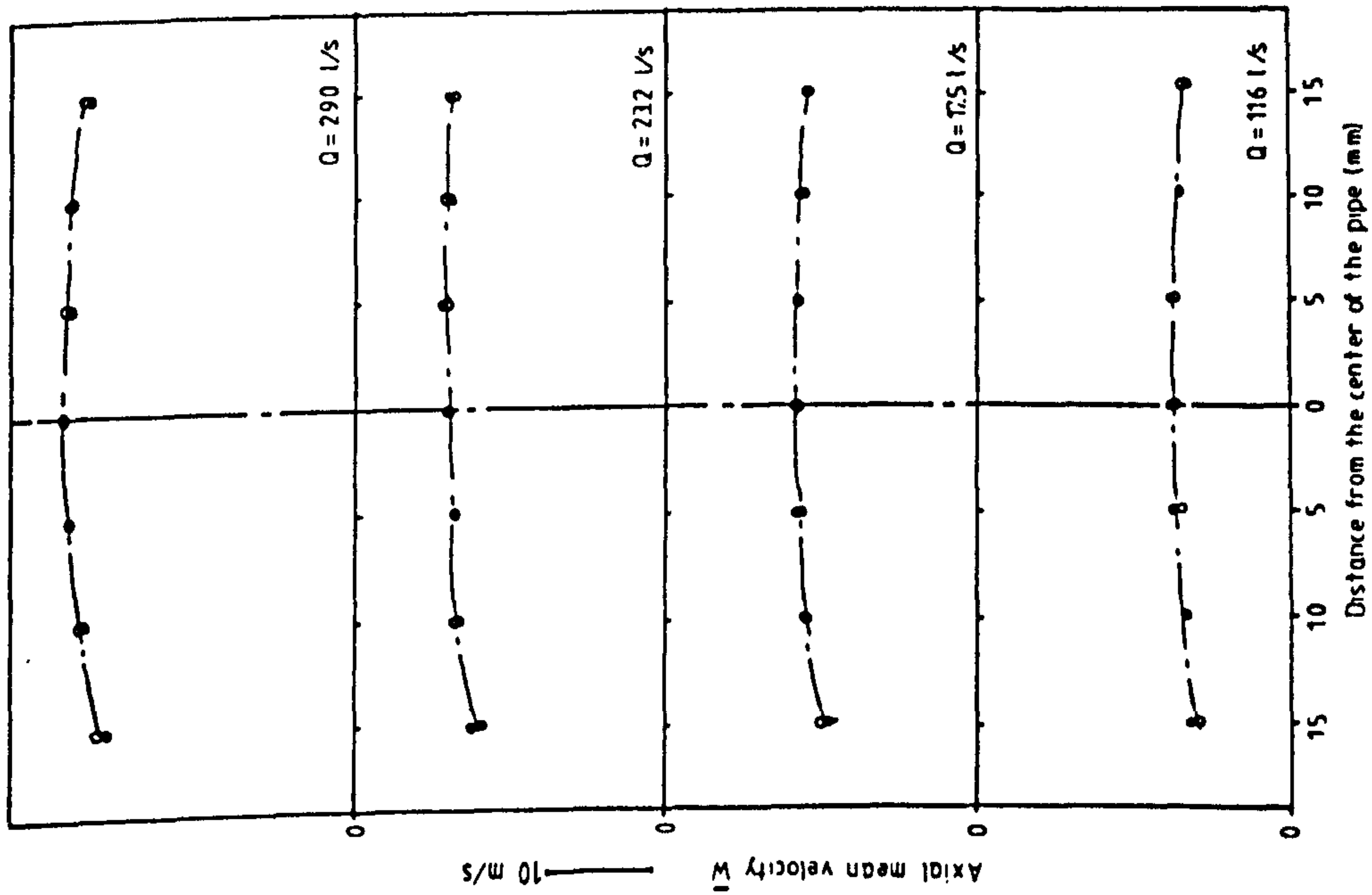
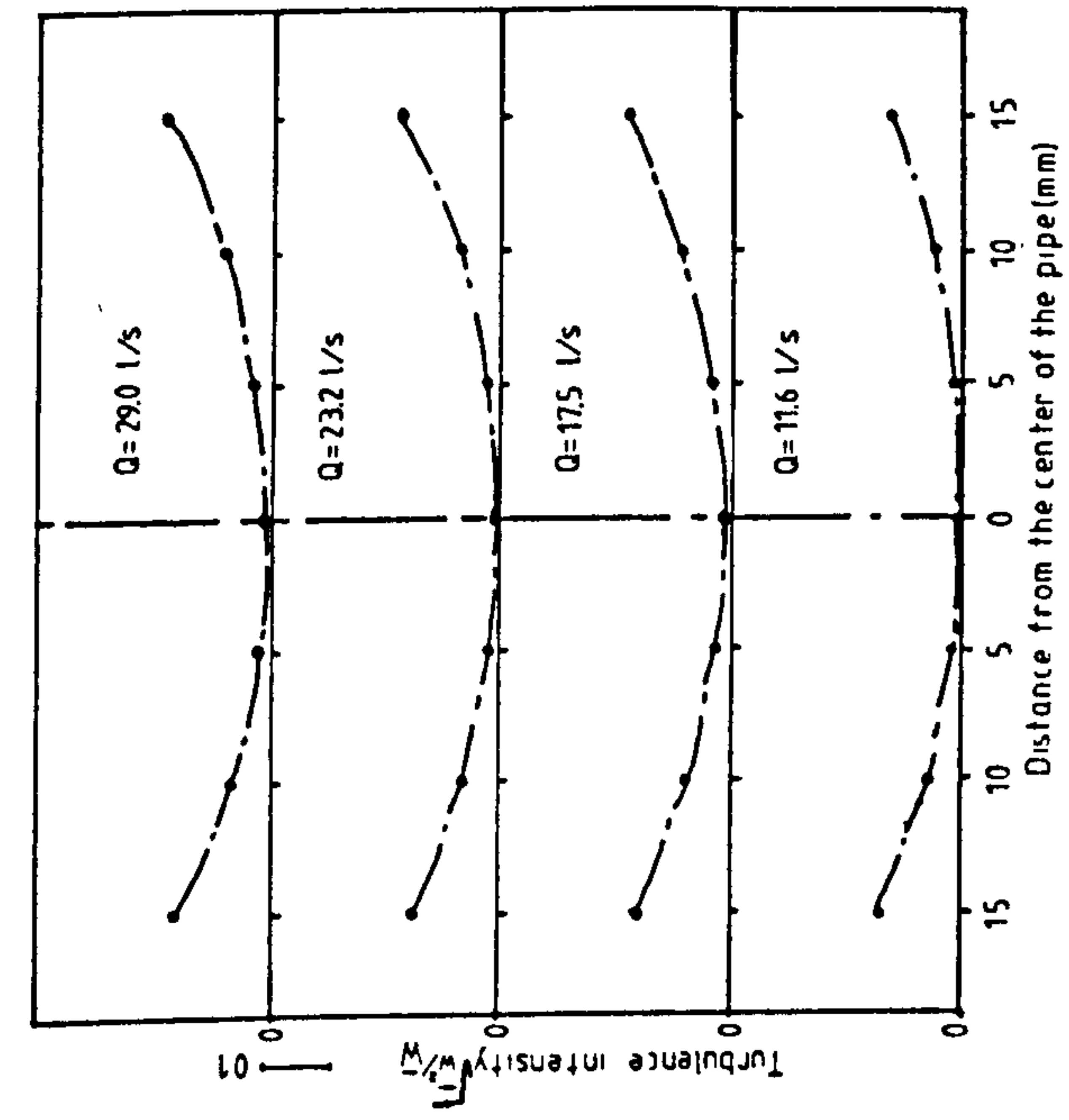


Fig. 5.9 Profiles of mean velocity and relative turbulence intensity at Plane 1 for the variable ramp throttling device.



Q = Volumetric flow rate through the system.

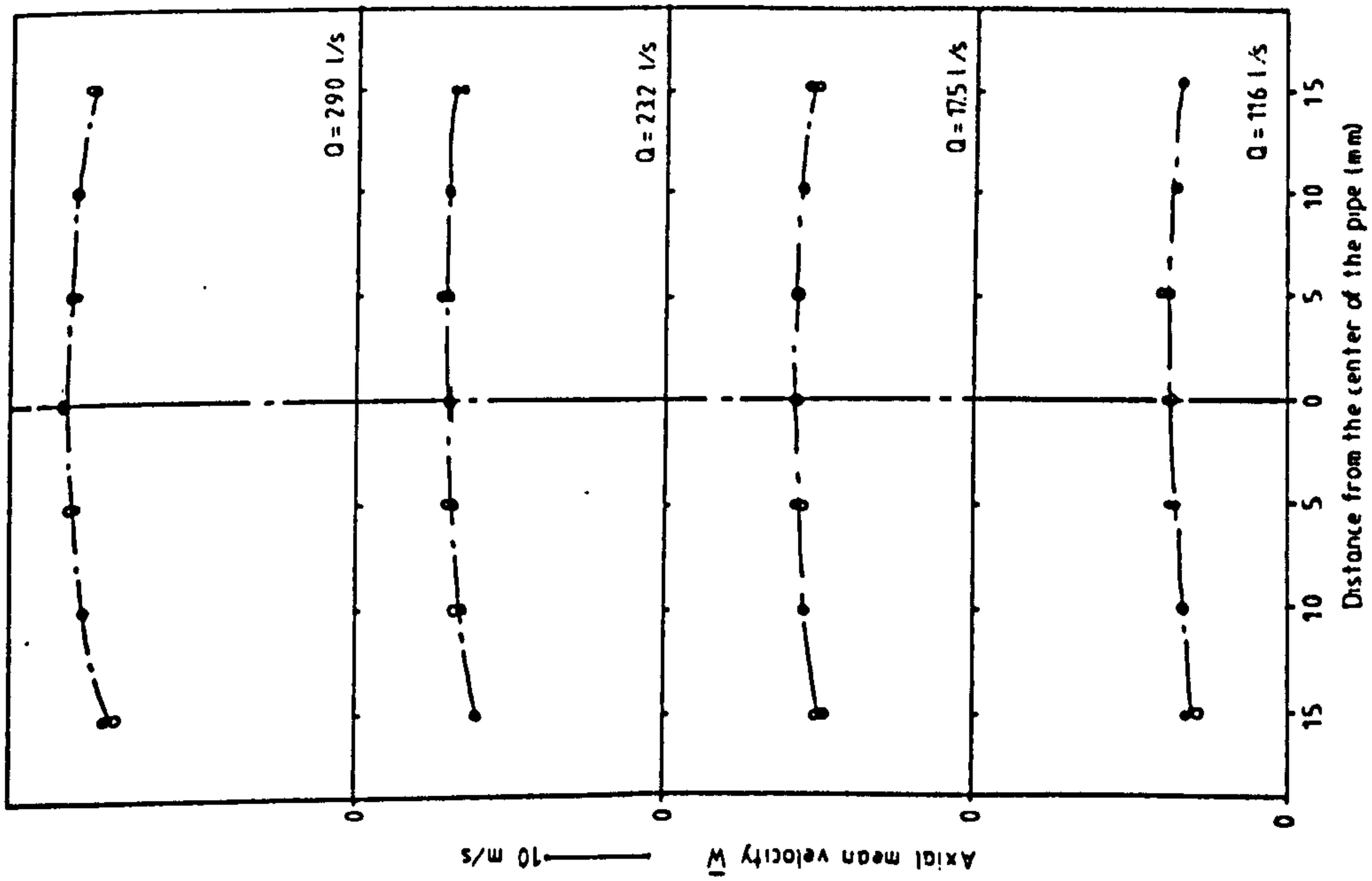


Fig. 5.10 Profiles of mean velocity and relative turbulence intensity at Plane 1 for the butterfly valve throttling device.

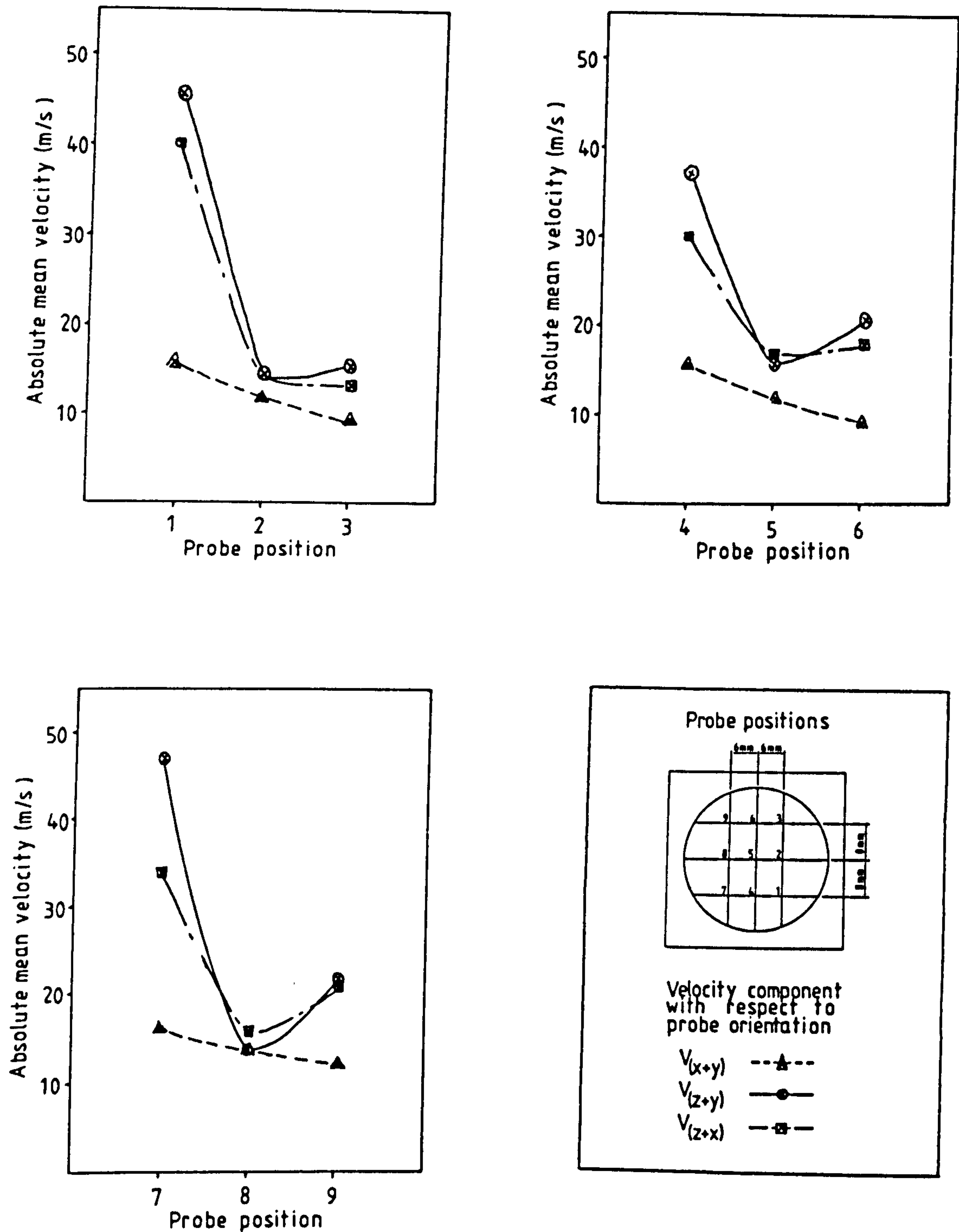


Fig. 5.11 Mean velocity measurements at Plane 2 for the variable ramp device at $\Theta = 110^\circ$.

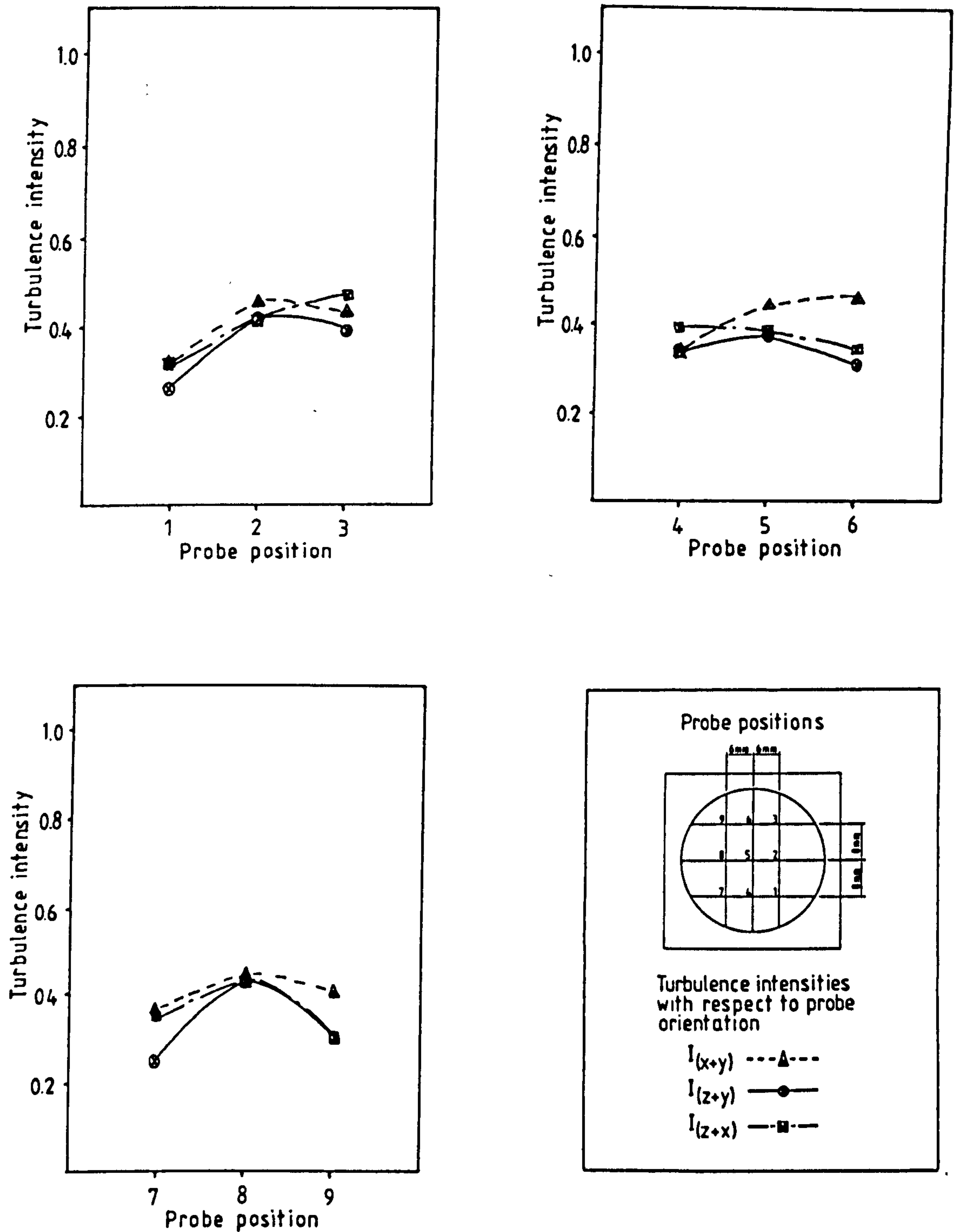


Fig. 5.12 Local turbulence intensity measurements ($\sqrt{v^2} / V$) for the variable ramp device at $\Theta = 110^\circ$.

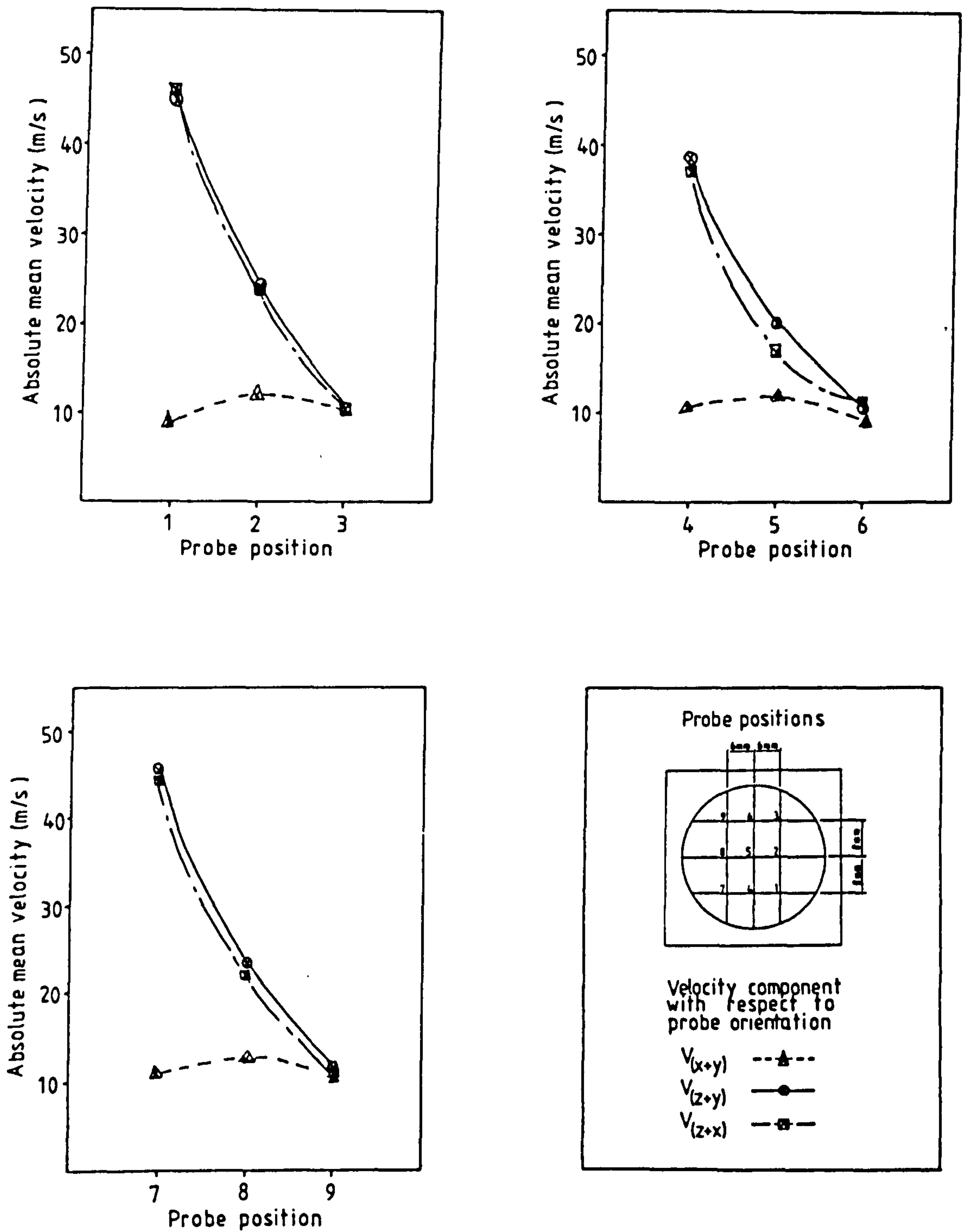


Fig. 5.13 Mean velocity measurements at Plane 2 for the variable ramp device at $\Theta = 130^\circ$.

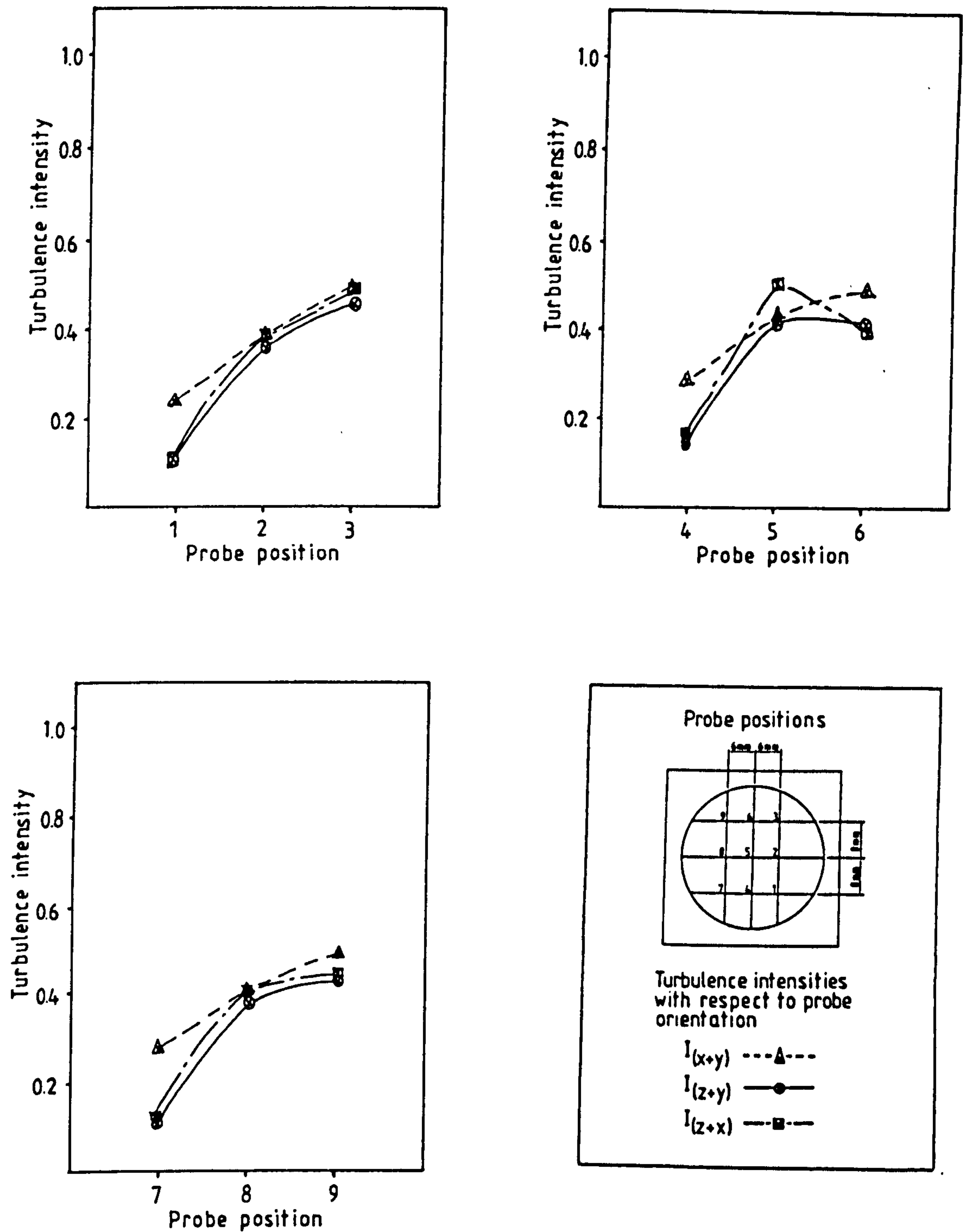


Fig. 5.14 Local turbulence intensity measurements ($\sqrt{v^2} / v$) for the variable ramp device at $\Theta = 130^\circ$.

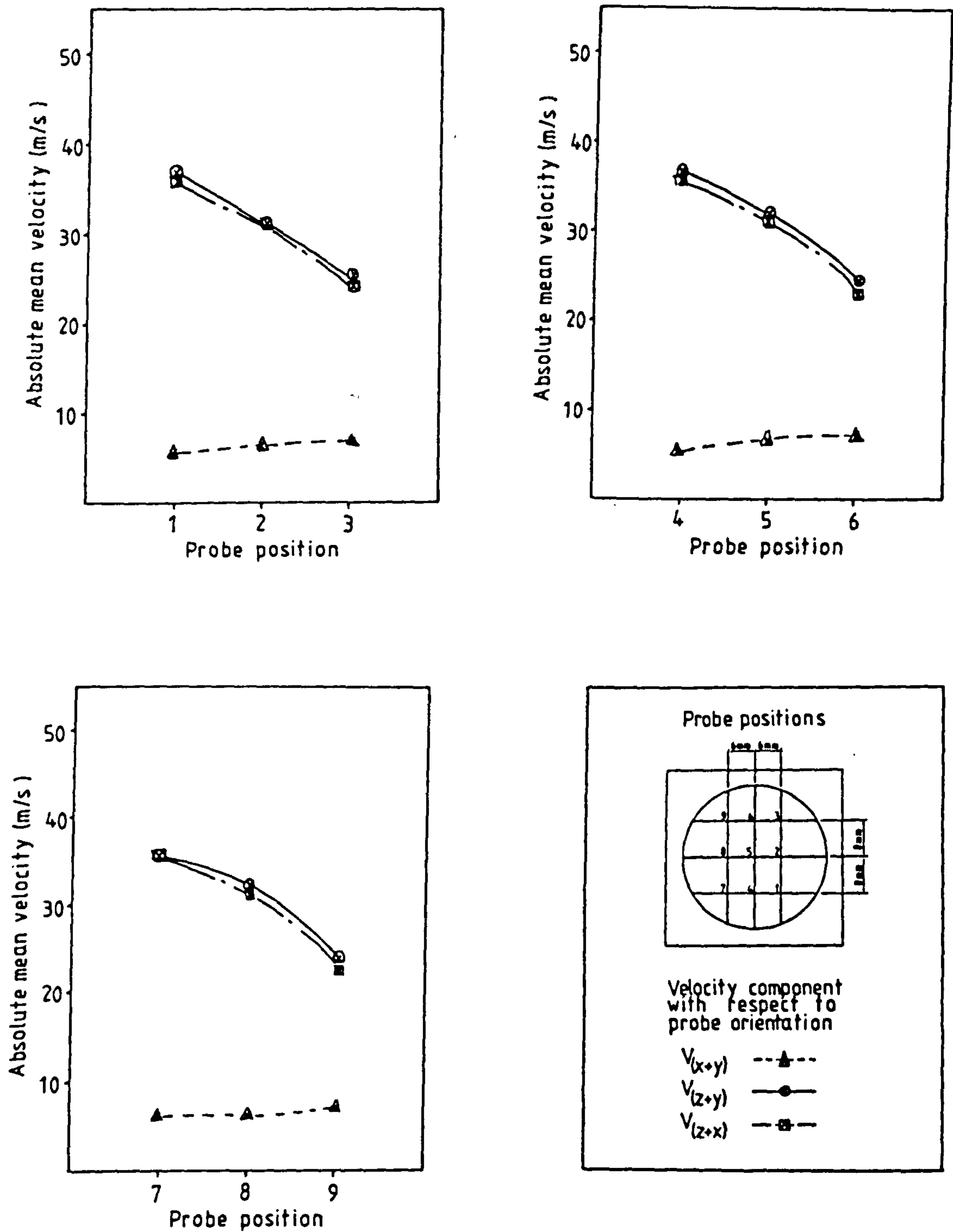


Fig. 5.15 Mean velocity measurements at Plane 2 for the variable ramp device at $\Theta = 150^\circ$.

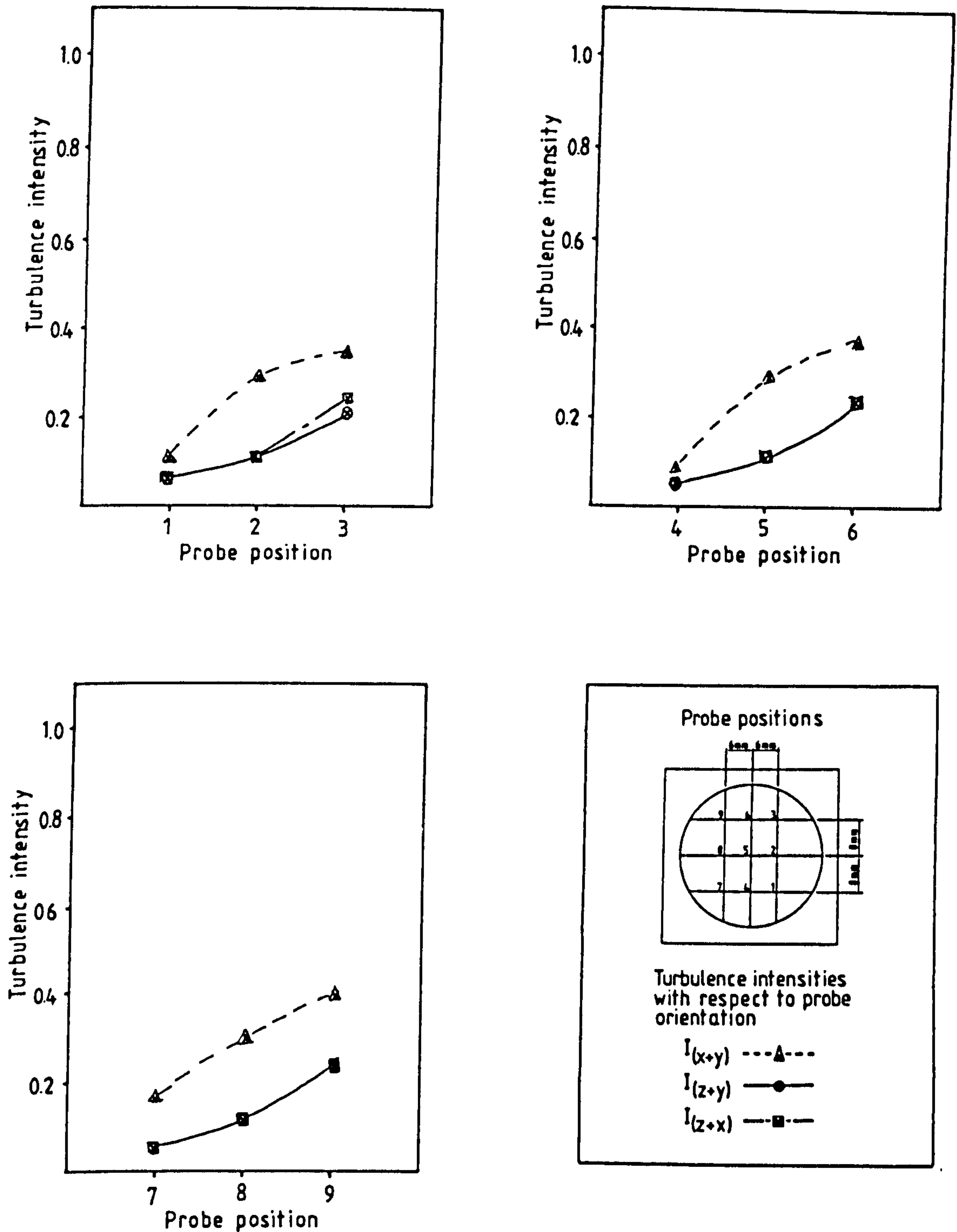


Fig. 5.16 Local turbulence intensity measurements ($\sqrt{v^2} / V$) for the variable ramp device at $\Theta = 150^\circ$.

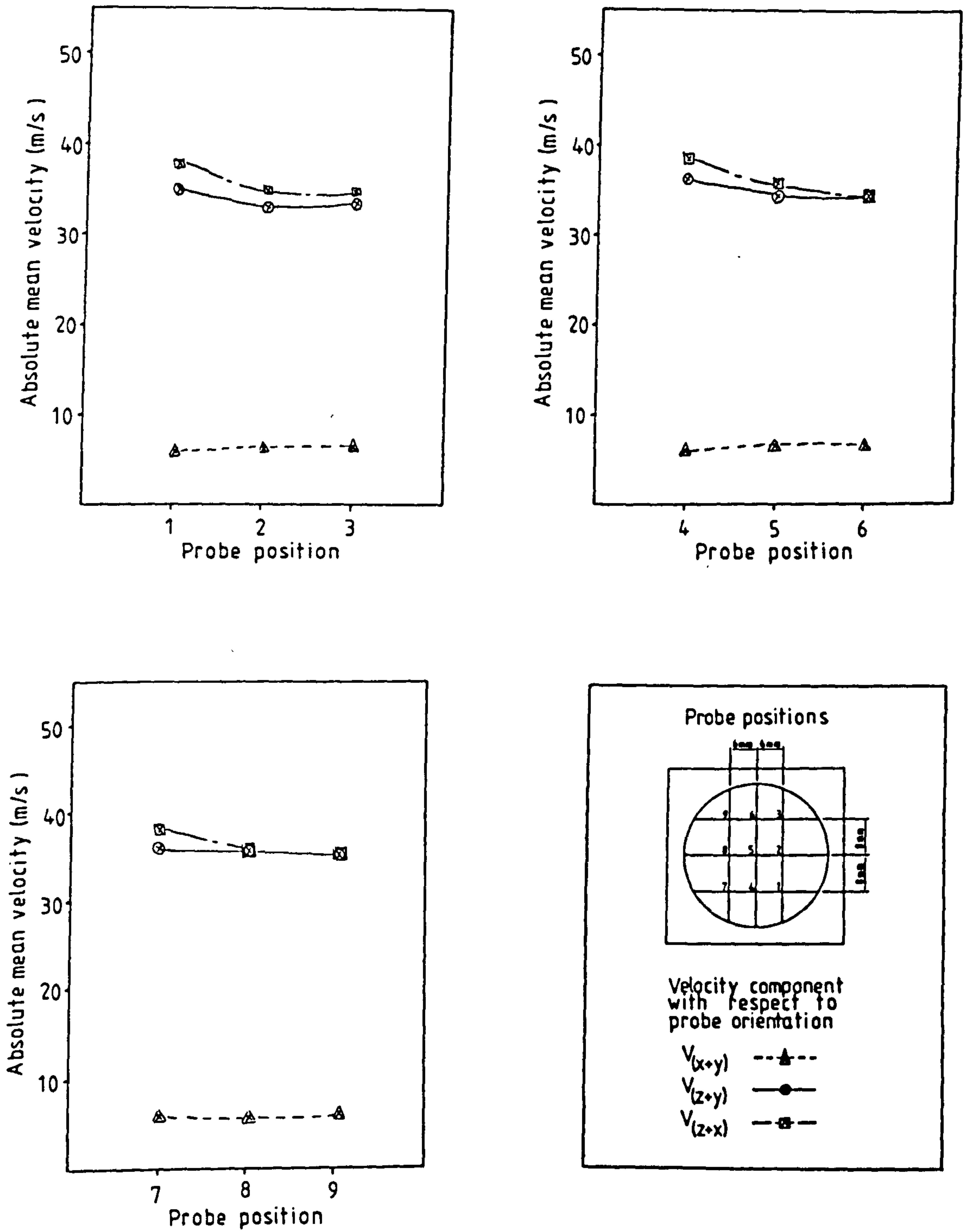


Fig. 5.17 Mean velocity measurements at Plane 2 for the variable ramp device at WOT.

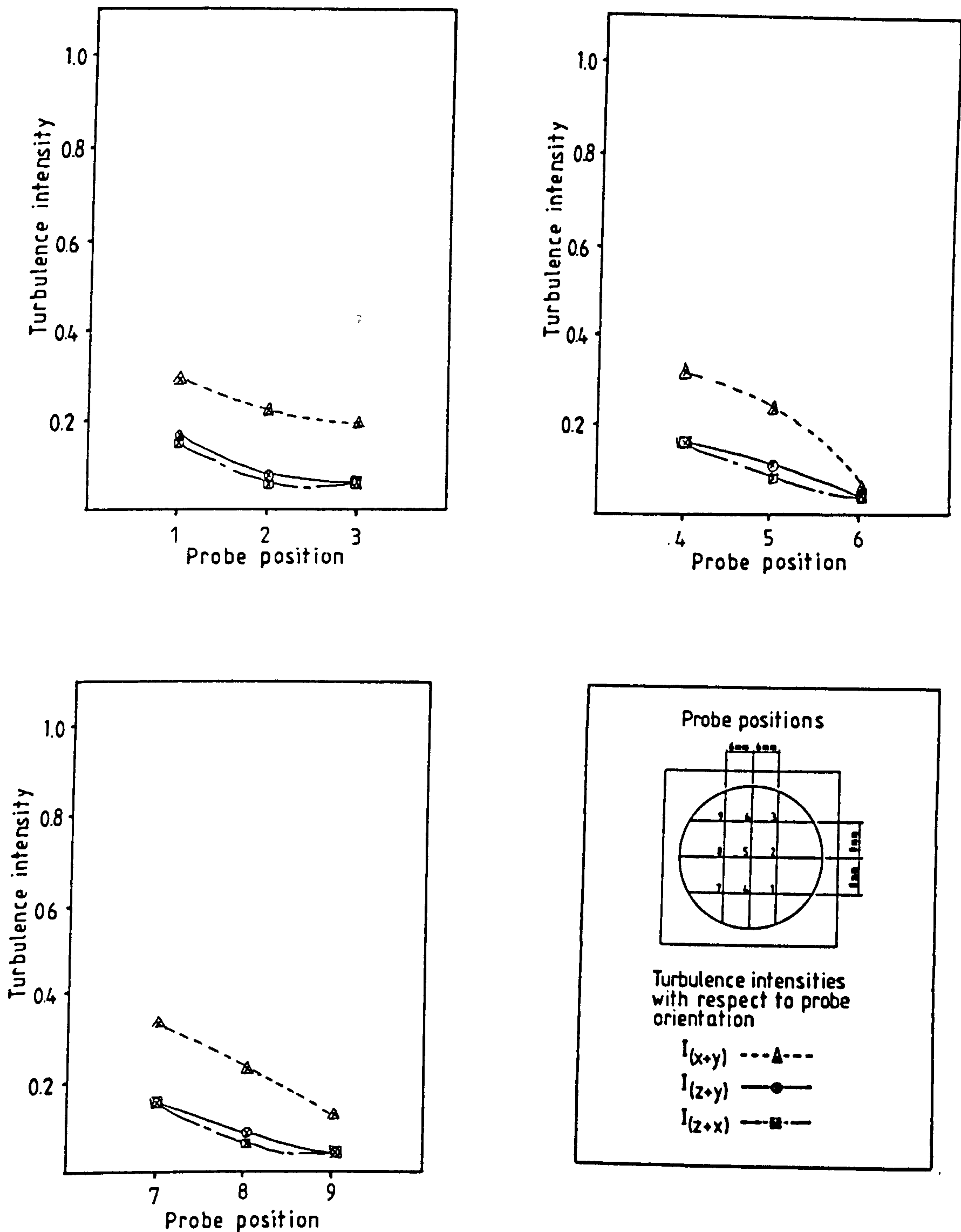


Fig. 5.18 Local turbulence intensity measurements ($\sqrt{v'^2} / V$) for the variable ramp device at WOT

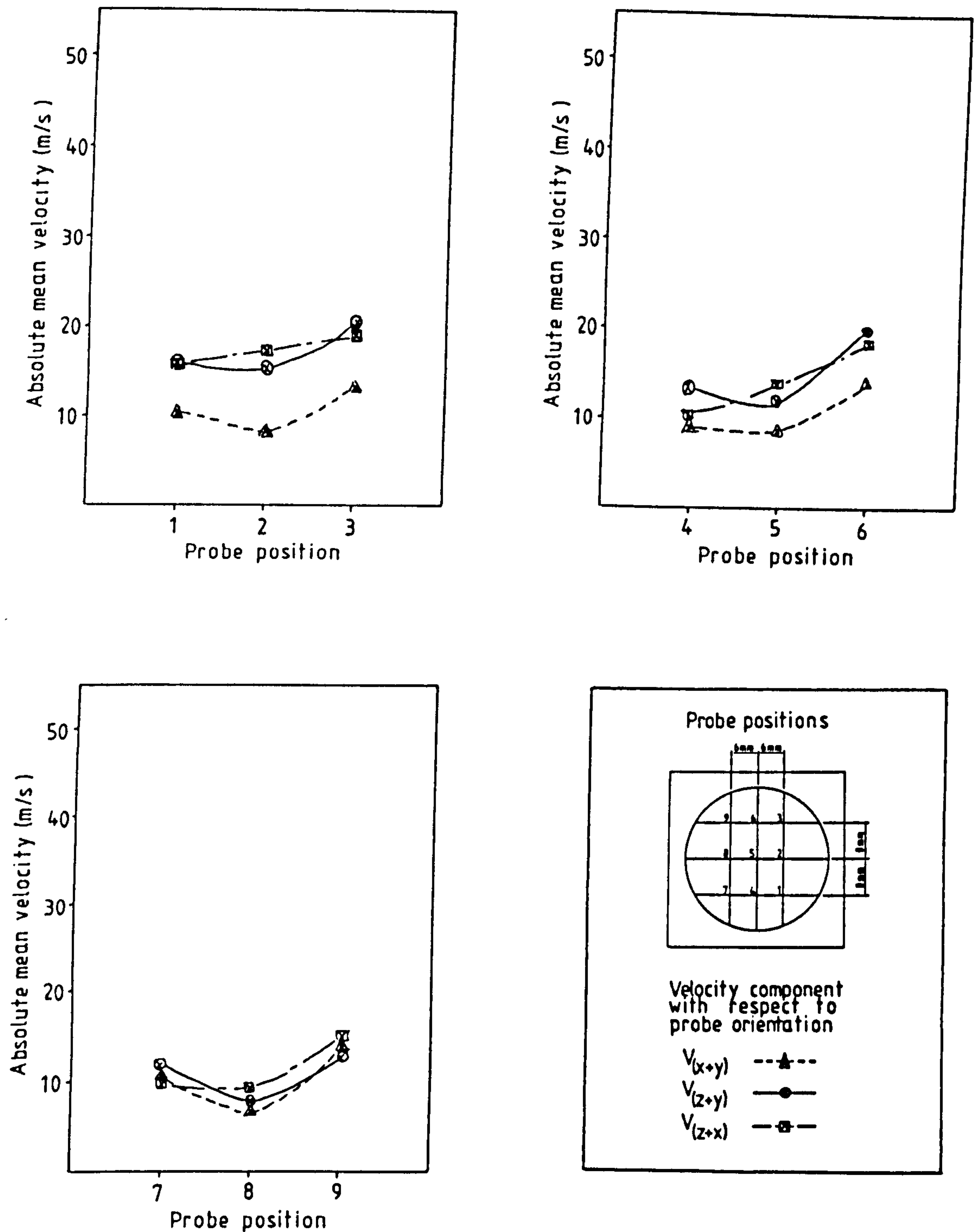


Fig. 5.19 Mean velocity measurements at Plane 2 for the butterfly valve device at $\beta = 30^\circ$.

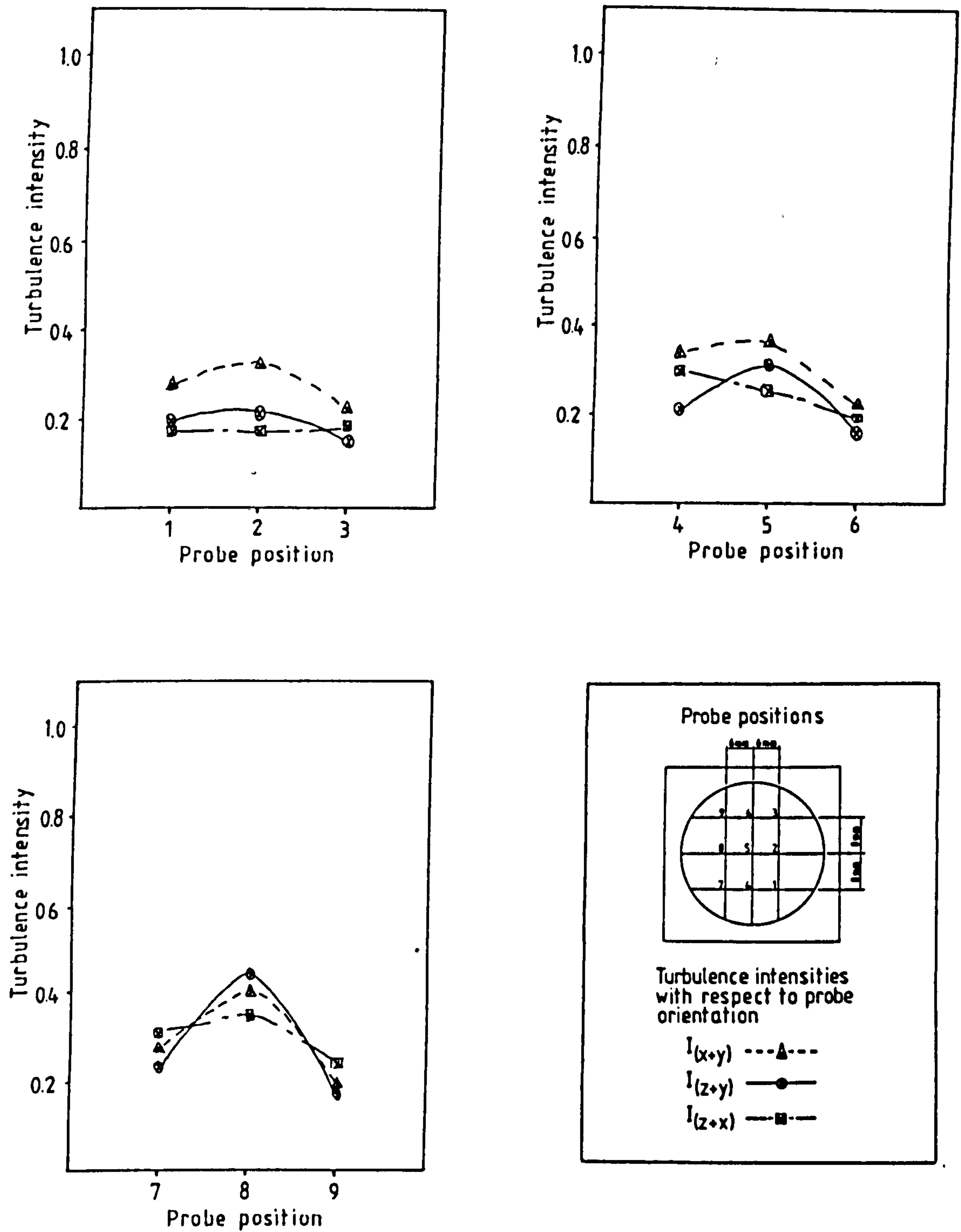


Fig. 5.20 Local turbulence intensity measurements ($\sqrt{v^2} / v$) for the butterfly valve at $\beta = 30^\circ$.

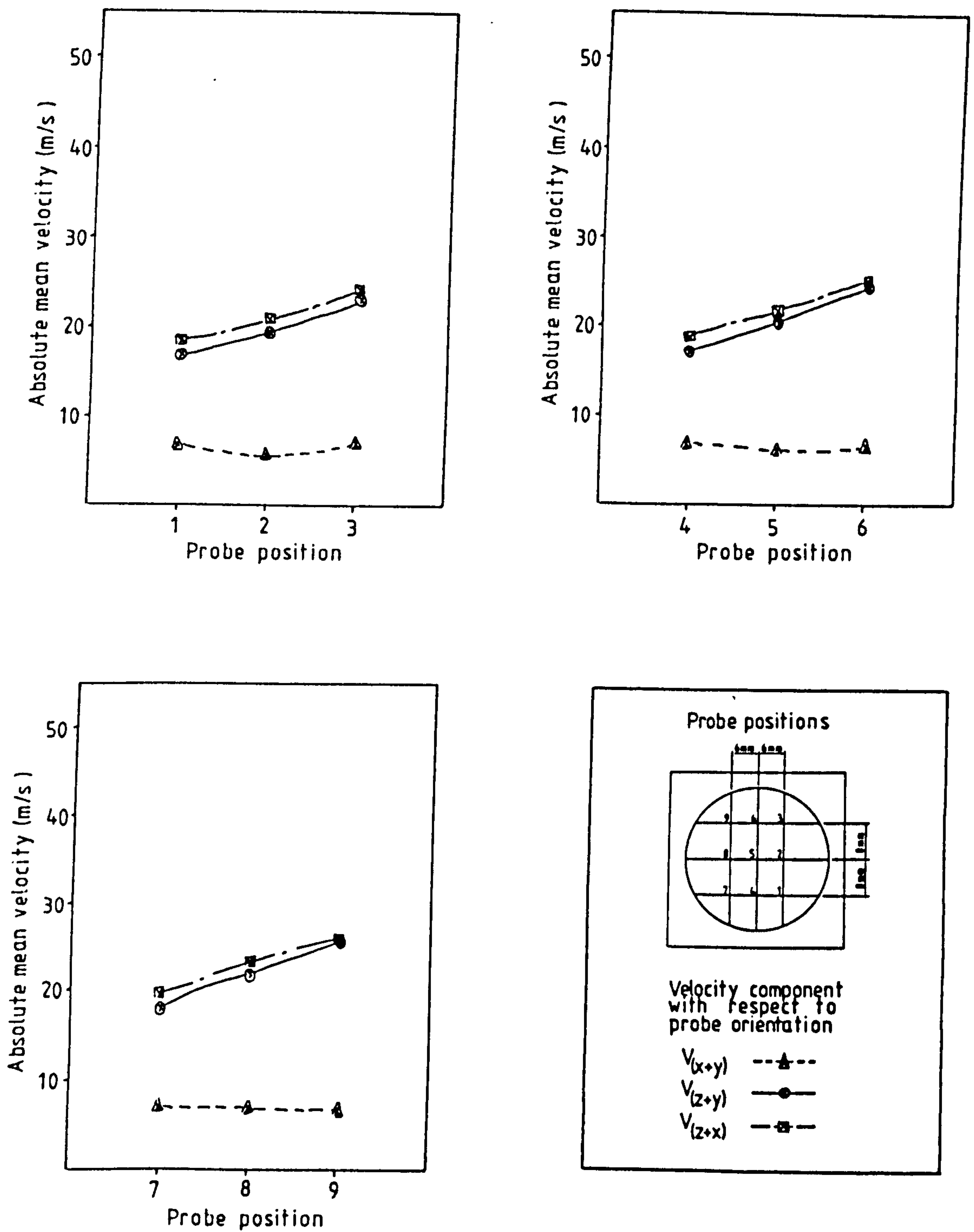


Fig. 5.21 Mean velocity measurements at Plane 2 for the butterfly valve device at $\beta = 50^\circ$.

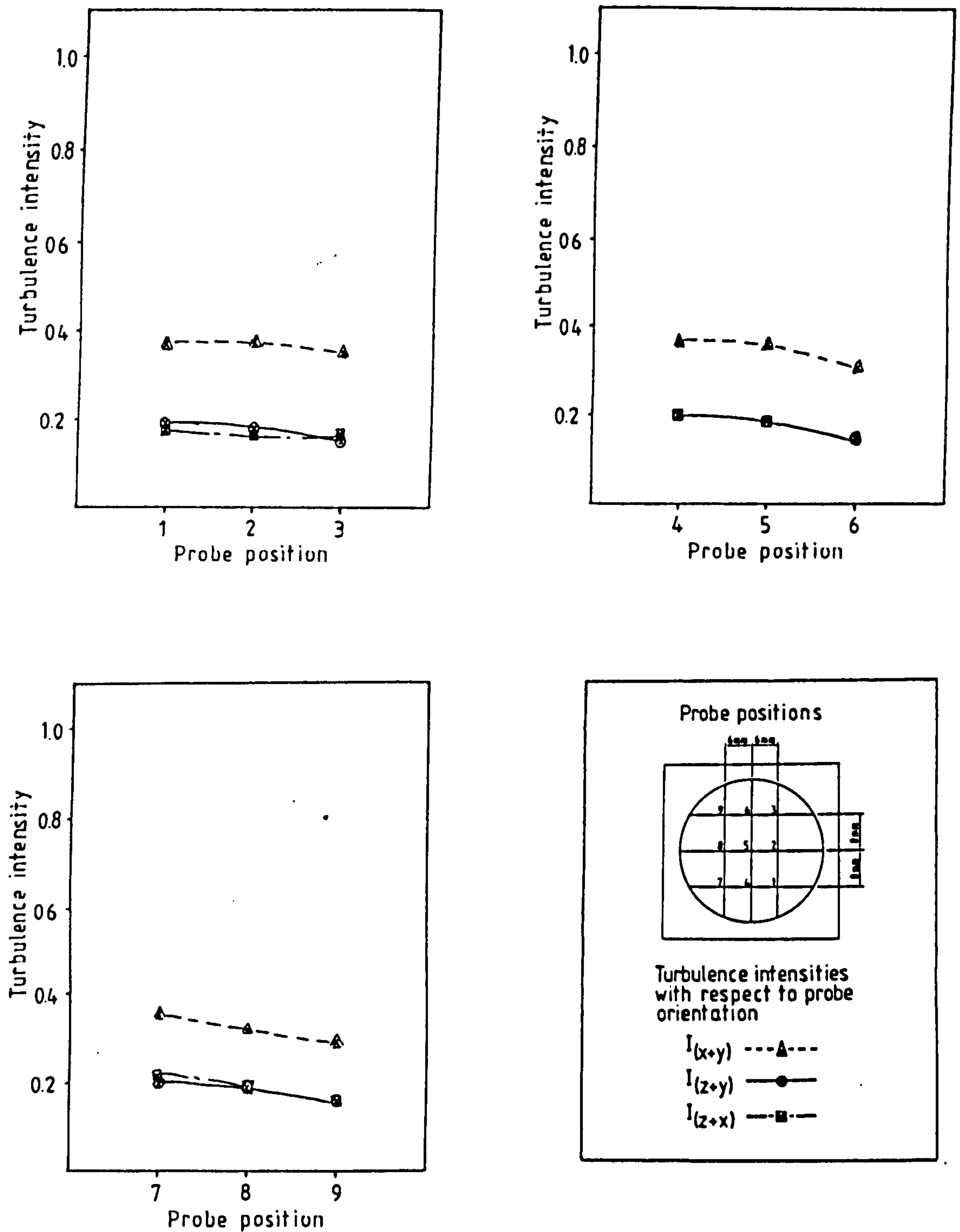


Fig. 5.22 Local turbulence intensity measurements ($\sqrt{v^2} / V$) for the butterfly valve at $\beta = 50^\circ$.

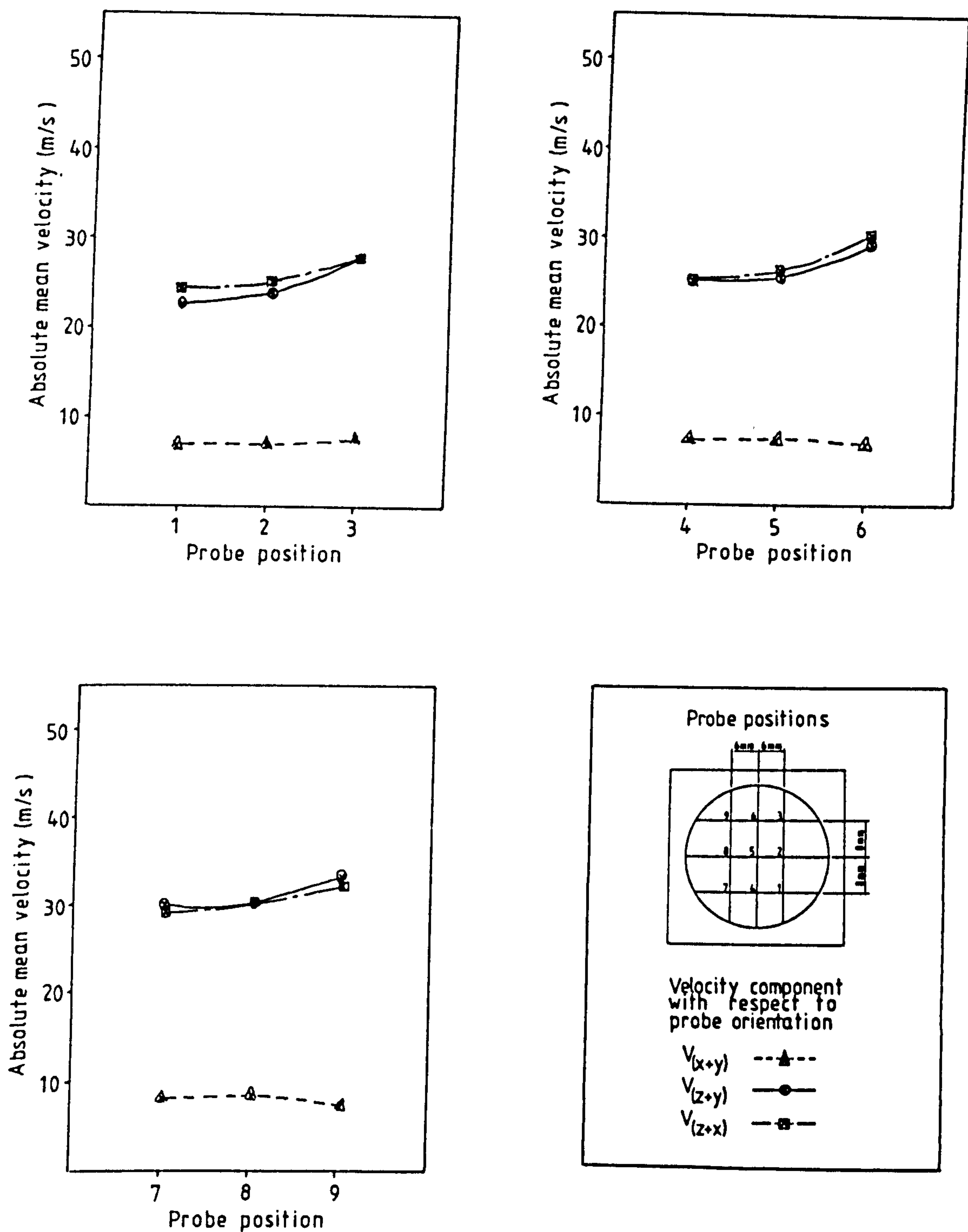


Fig. 5.23 Mean velocity measurements at Plane 2 for the butterfly valve device at $\beta = 65^\circ$.

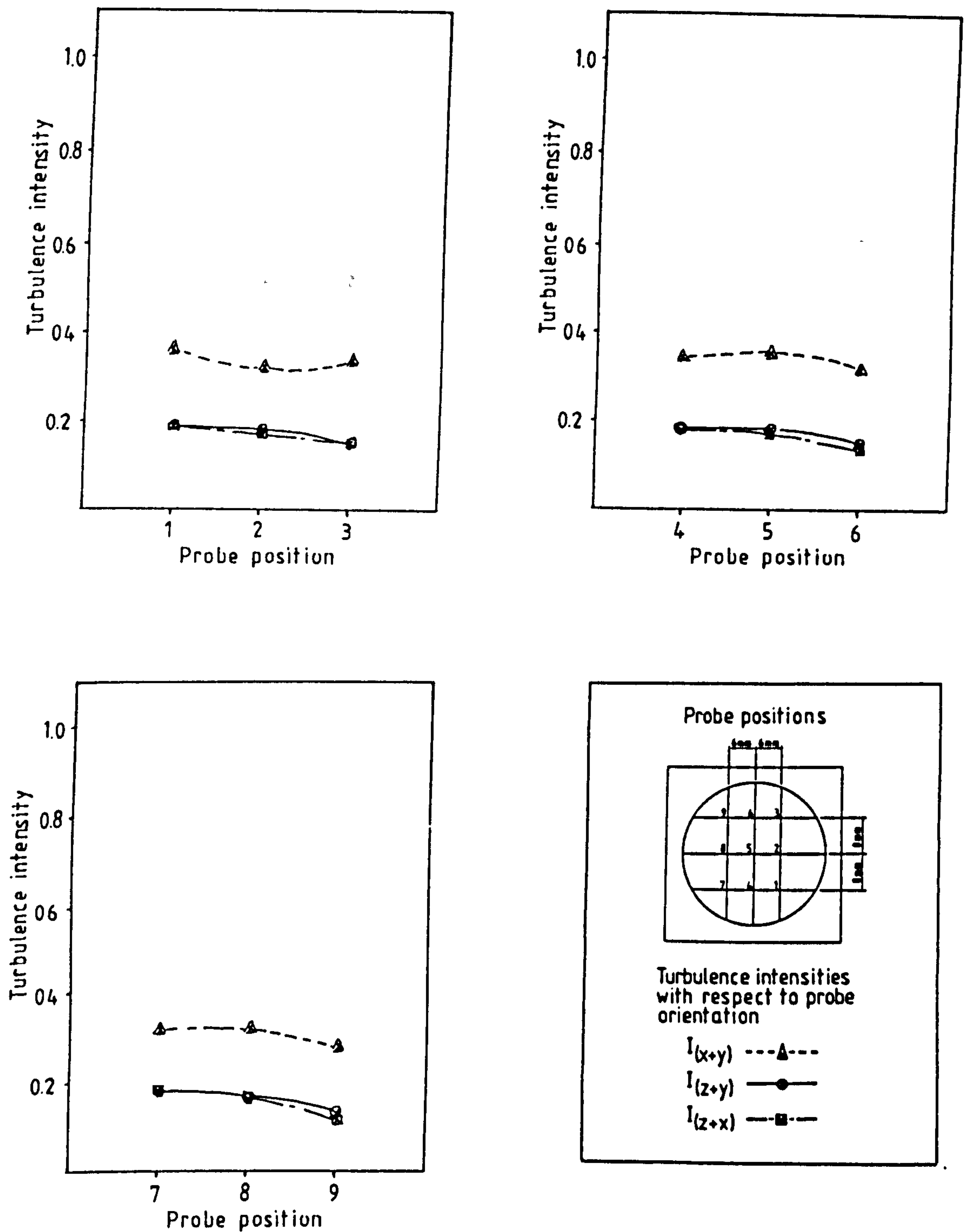


Fig. 5.24 Local turbulence intensity measurements ($\sqrt{v^2} / V$) for the butterfly valve at $\beta = 65^\circ$.

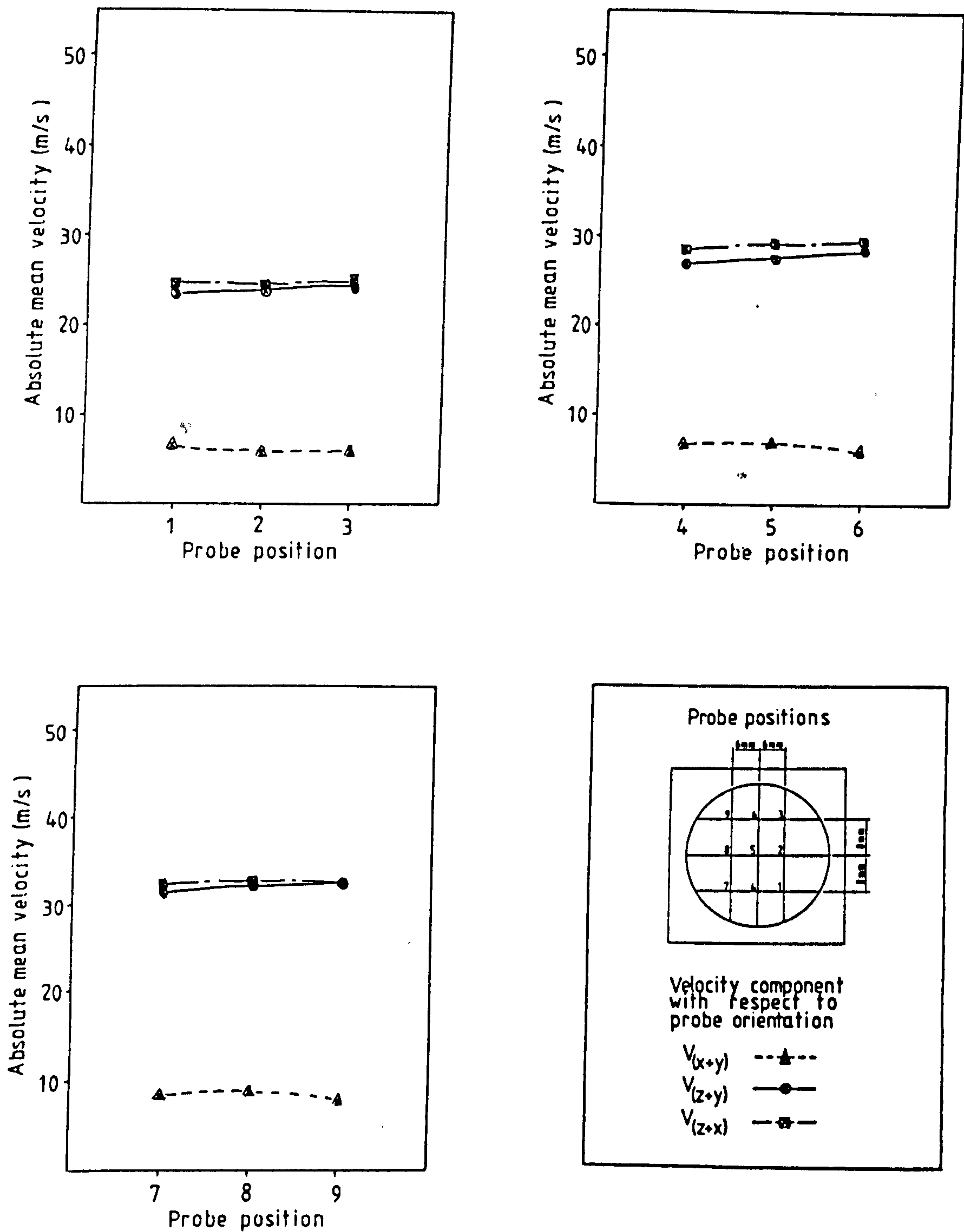


Fig. 5.25 Mean velocity measurements at Plane 2 for the butterfly valve device at $\beta = 75^\circ$.

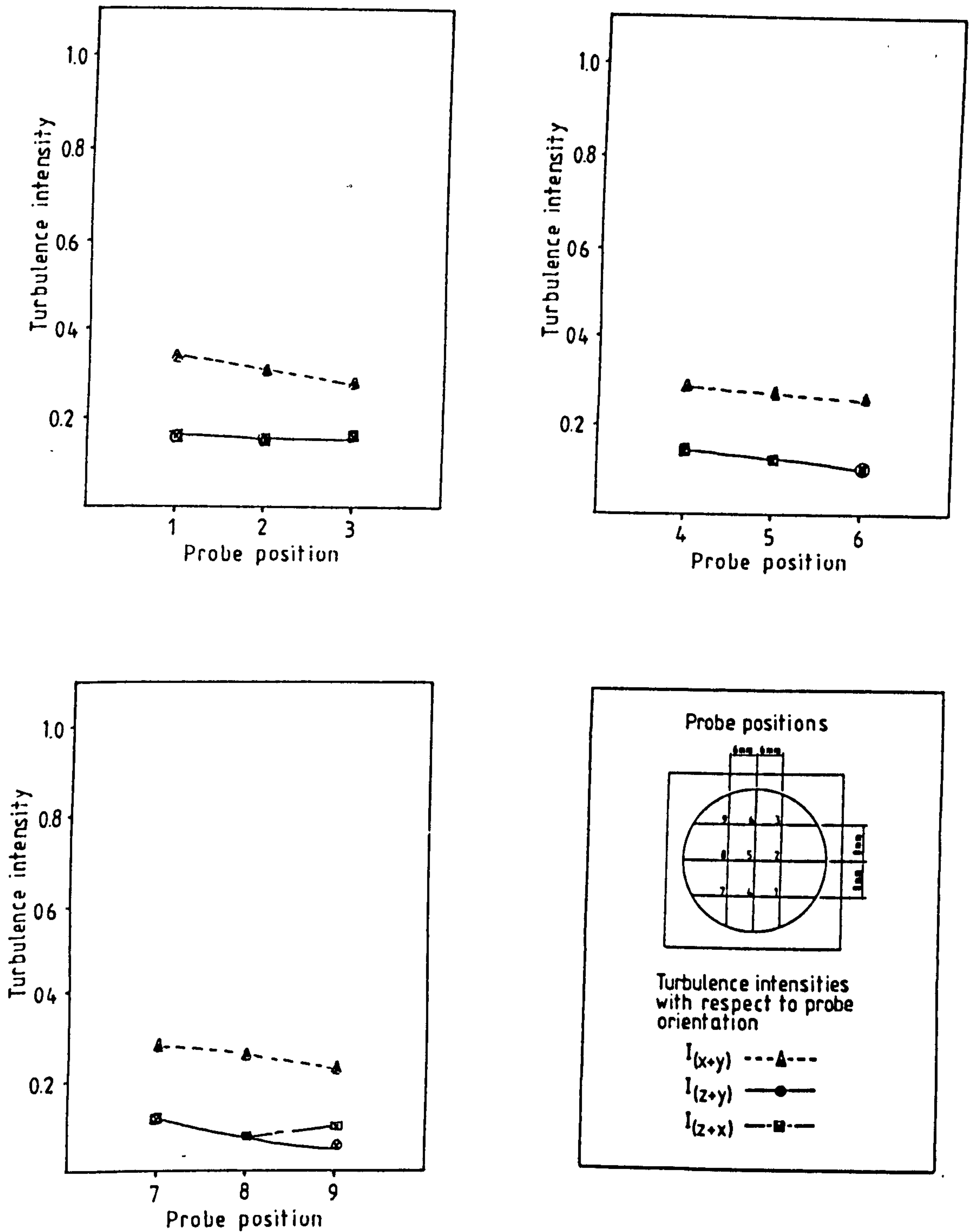


Fig. 5.26 Local turbulence intensity measurements ($\sqrt{v^2} / V$) for the butterfly valve at $\beta = 75^\circ$.

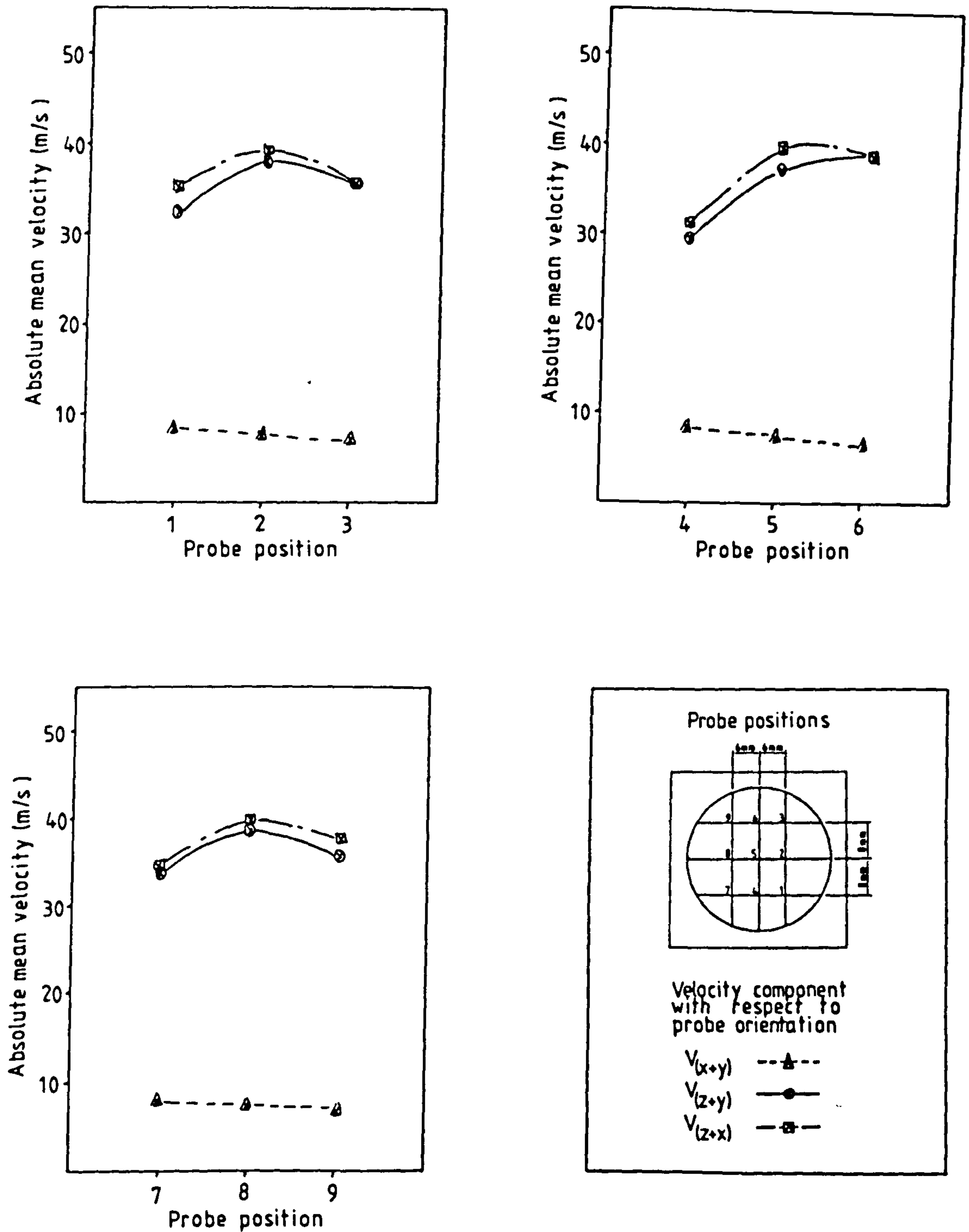


Fig. 5.27 Mean velocity measurements at Plane 2 for the butterfly valve device at WOT.

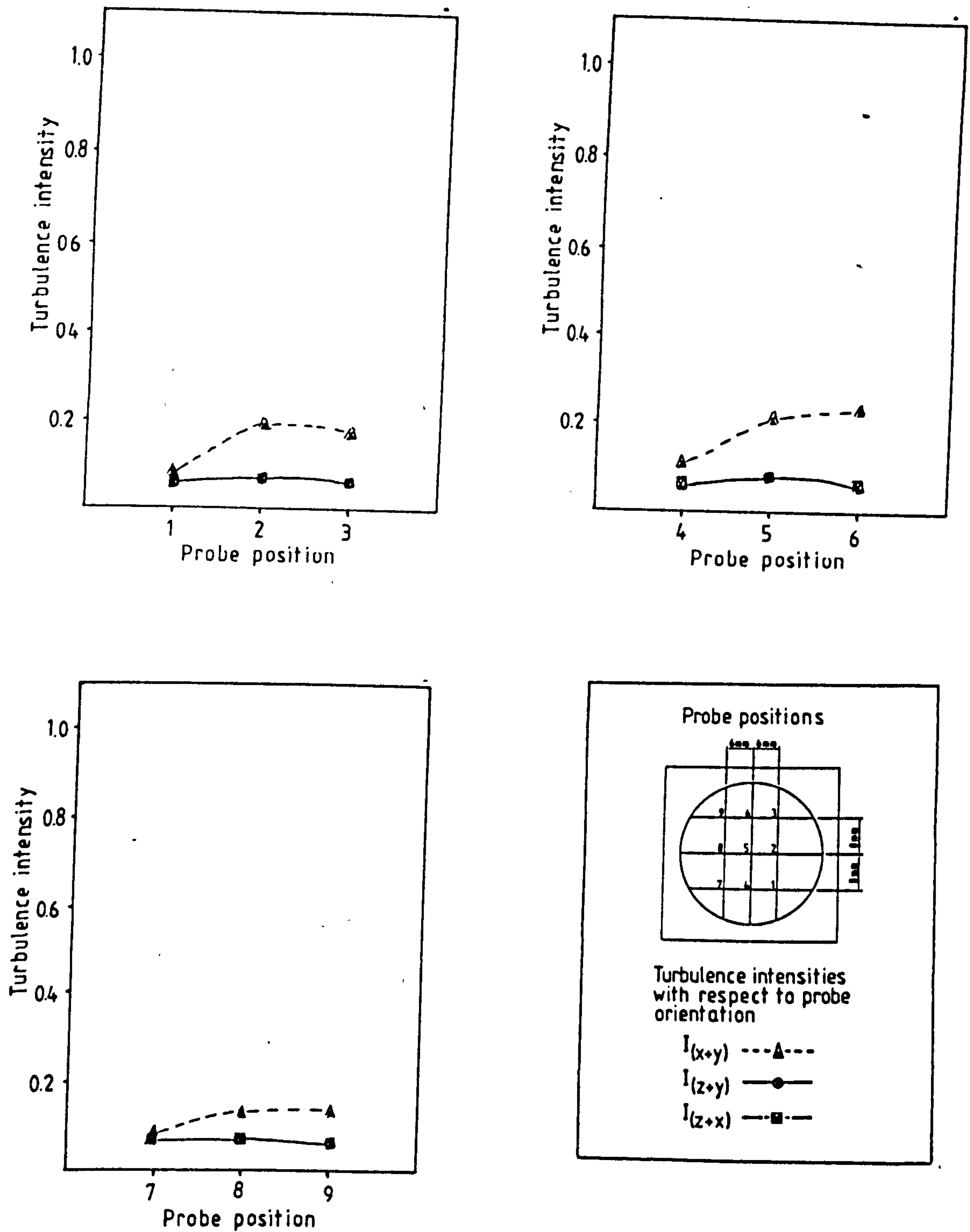


Fig. 5.28 Local turbulence intensity measurements ($\sqrt{\overline{v^2}} / v$) for the butterfly valve at WOT.

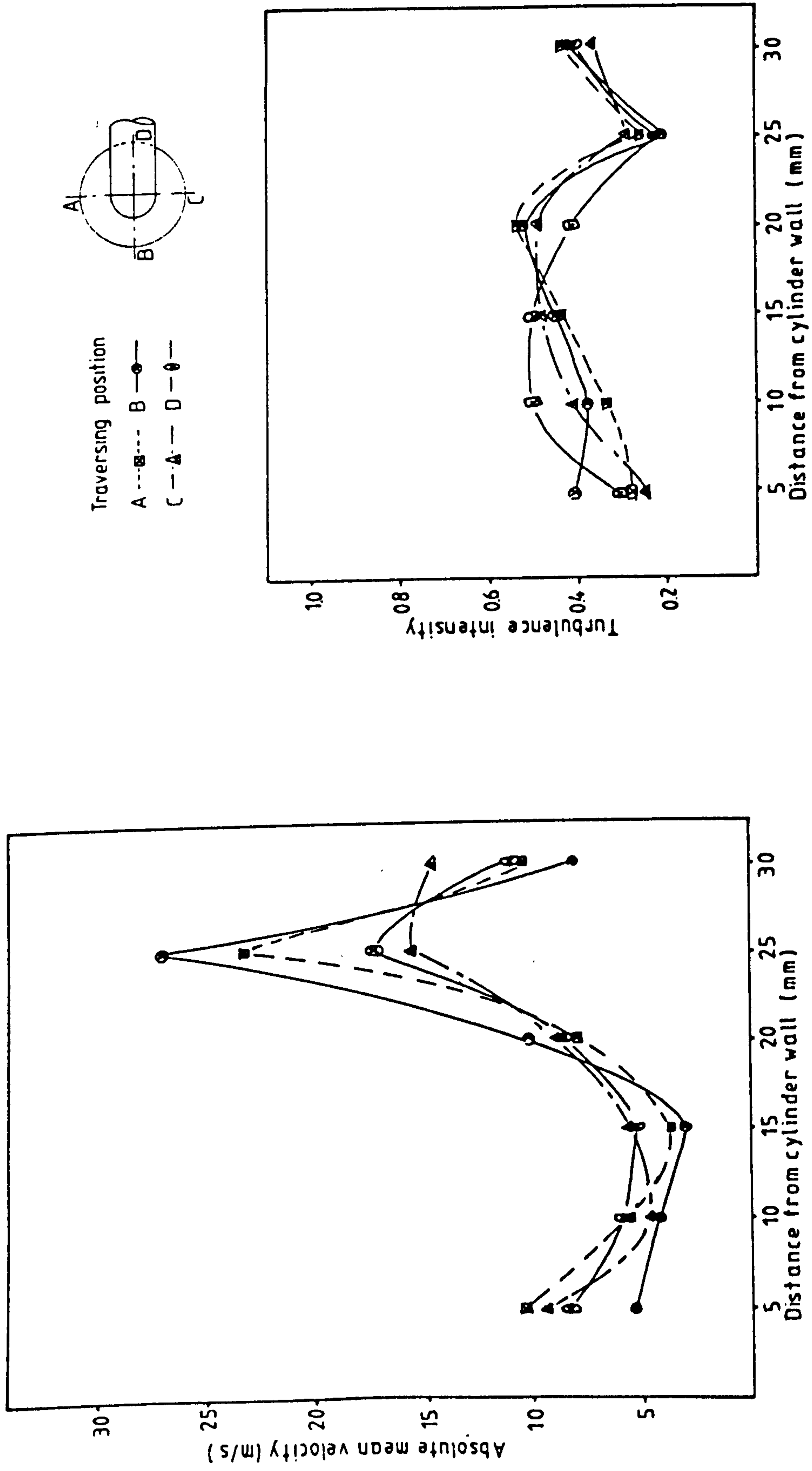


Fig. 5.29 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the variable ramp device at $\Theta = 110^\circ$. Wire orientation normal to (z, r) plane.

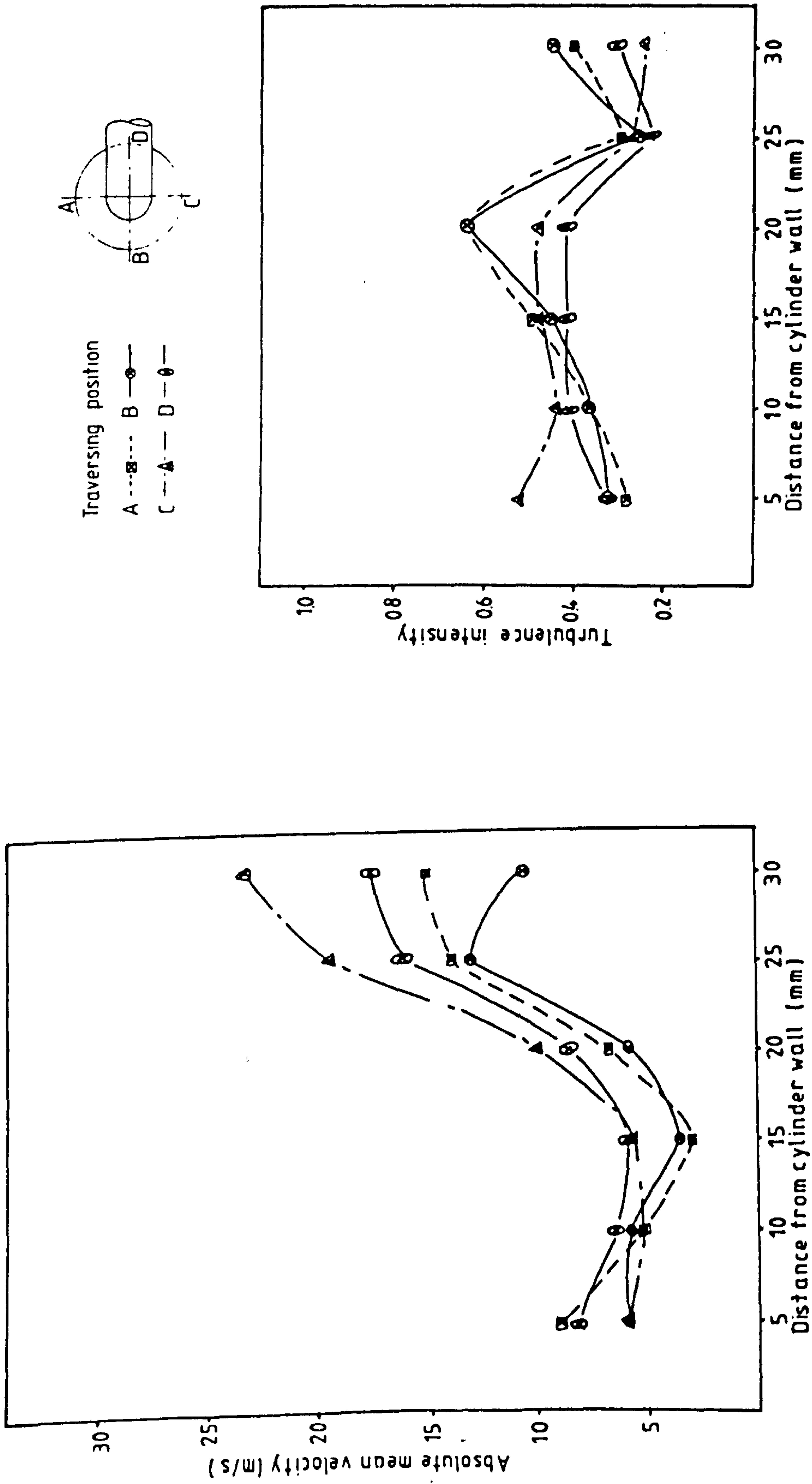


Fig. 5.30 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the variable ramp device at $\Theta = 110^\circ$. Wire orientation normal to (r, Θ) plane.

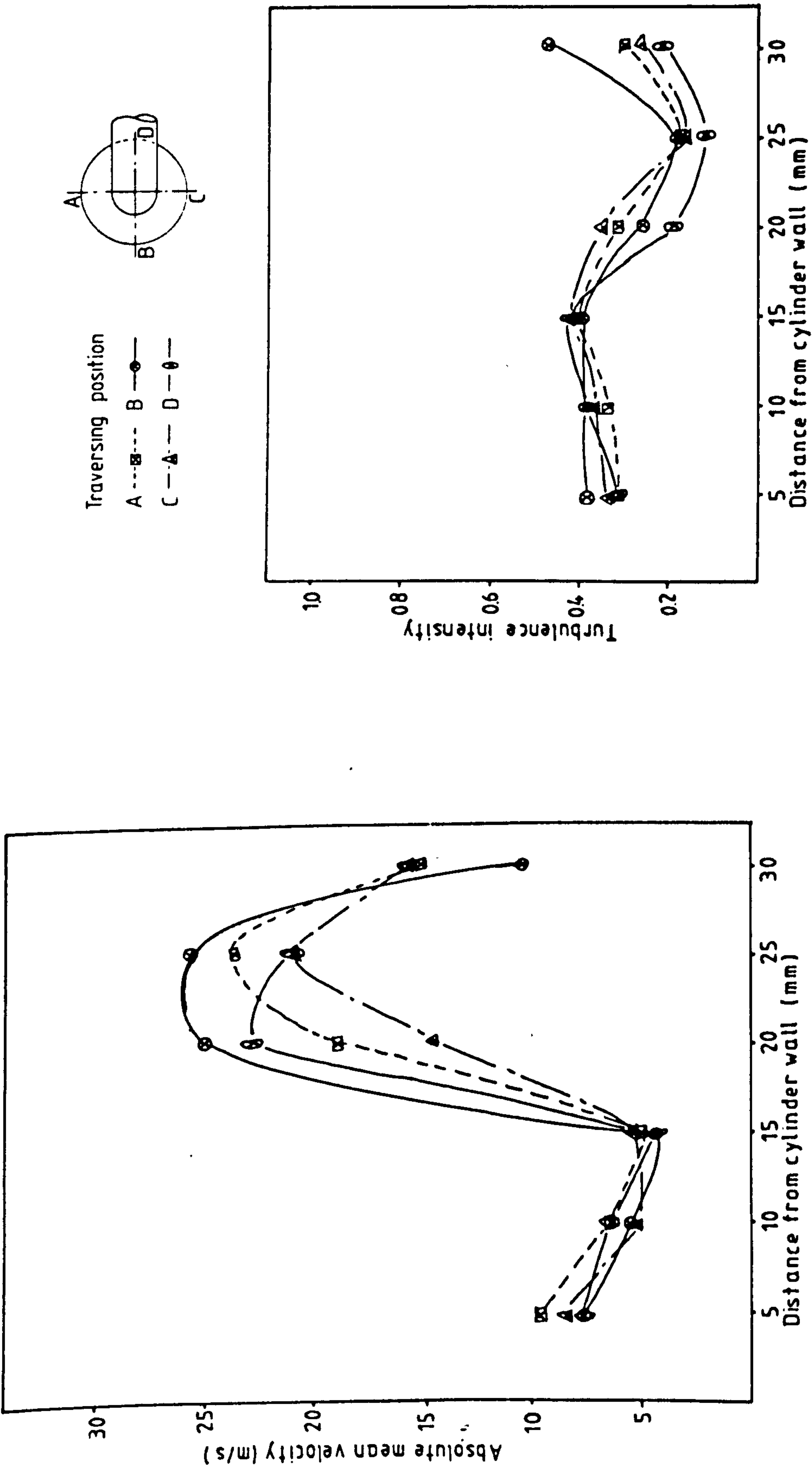


Fig. 5.31 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the variable ramp device at $\Theta = 130^\circ$. Wire orientation normal to (z, r) plane.

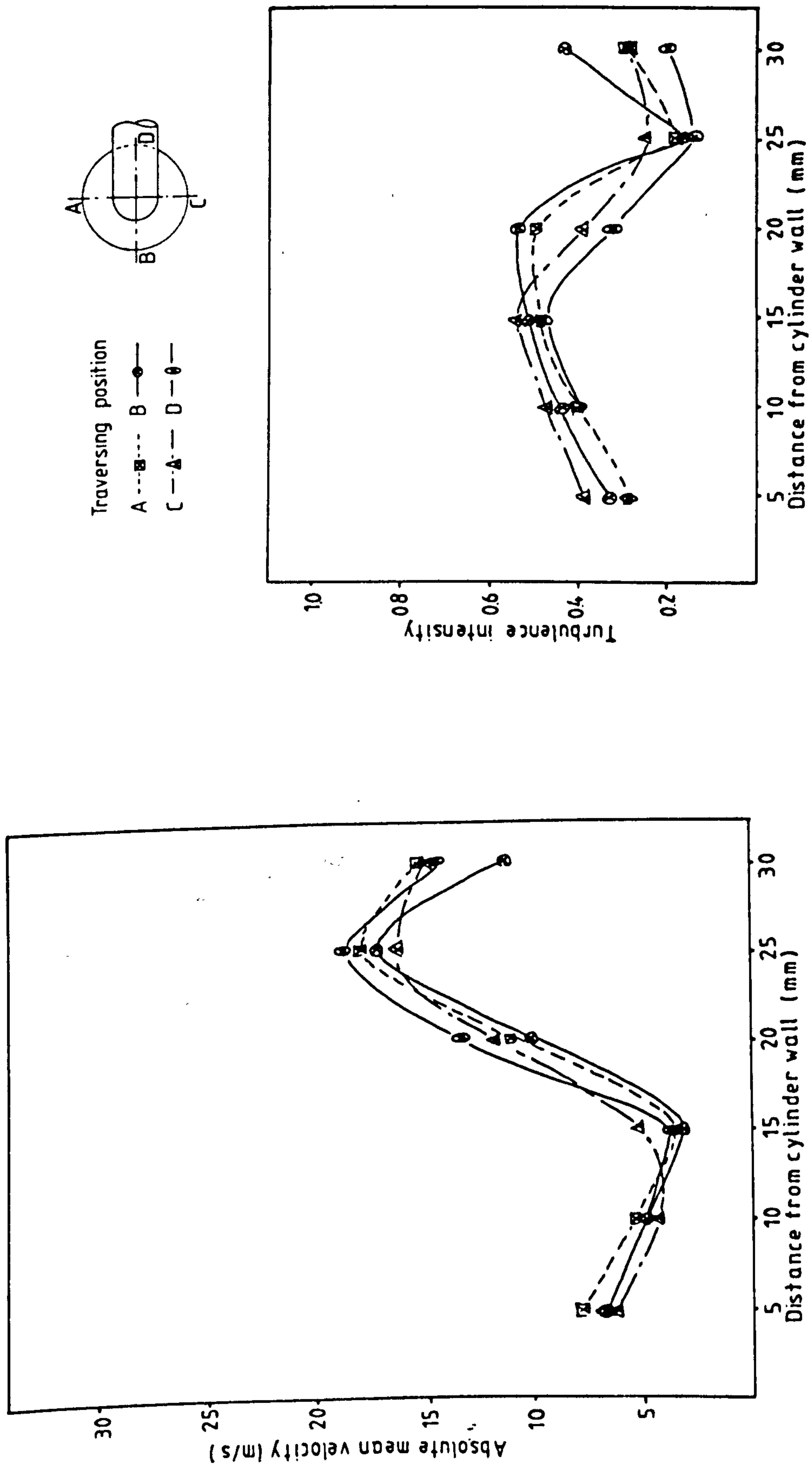


Fig. 5.32 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the variable ramp device at $\Theta = 130^\circ$. Wire orientation normal to (r, Θ) plane.

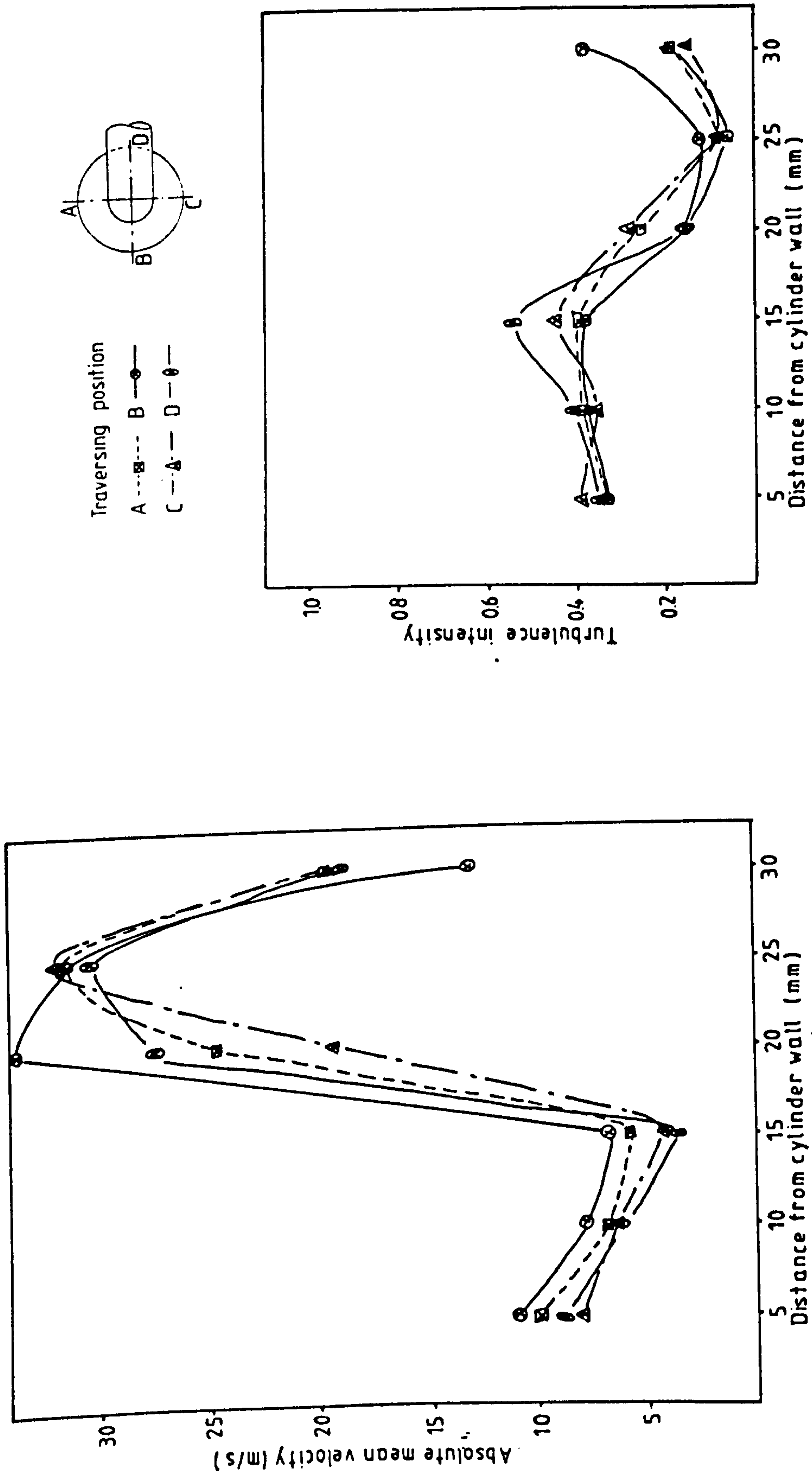


Fig. 5.33 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the variable ramp device at $\Theta = 150^\circ$. Wire orientation normal to (z, r) plane.

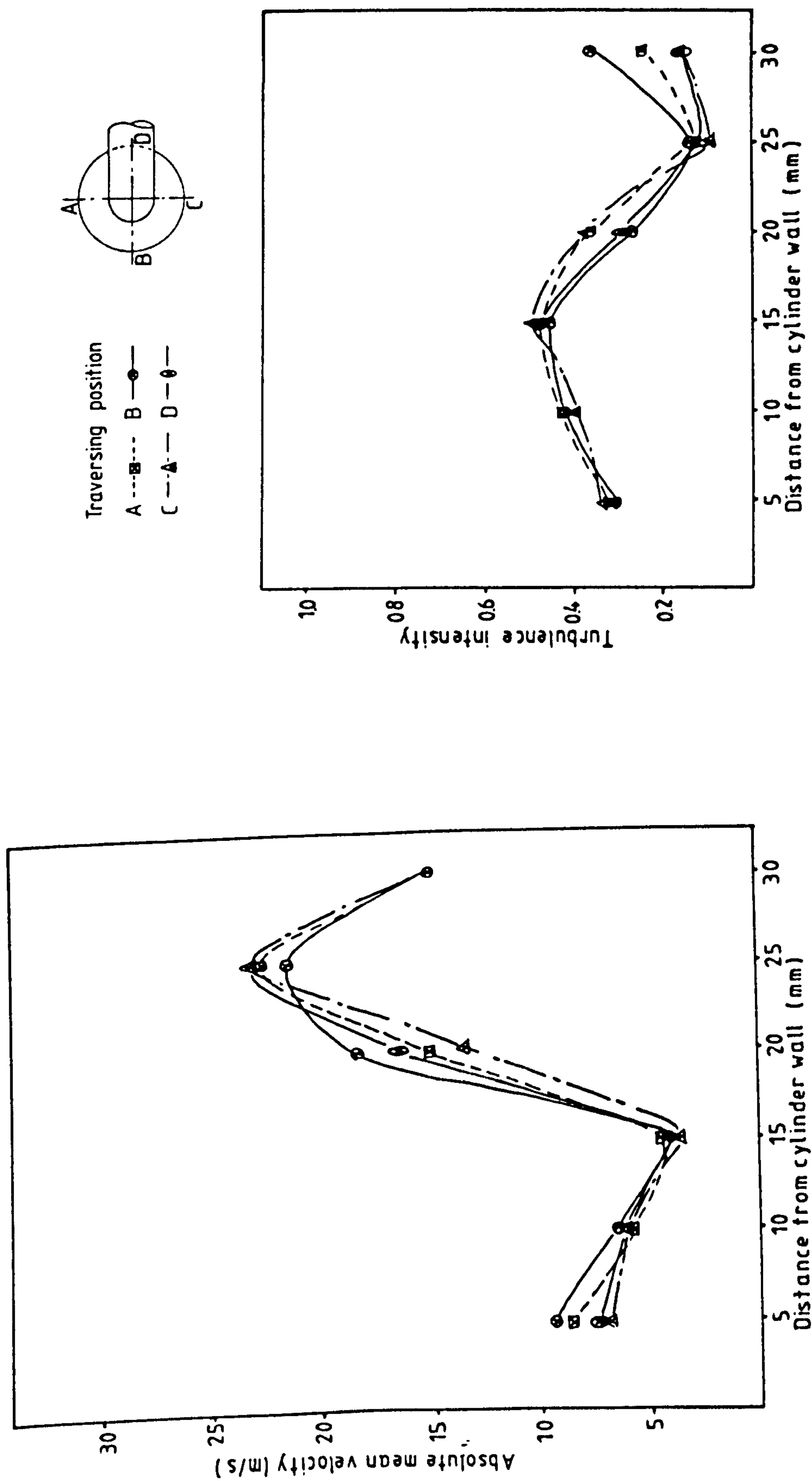


Fig. 5.34 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the variable ramp device at $\Theta = 150^\circ$. Wire orientation normal to (r, Θ) plane.

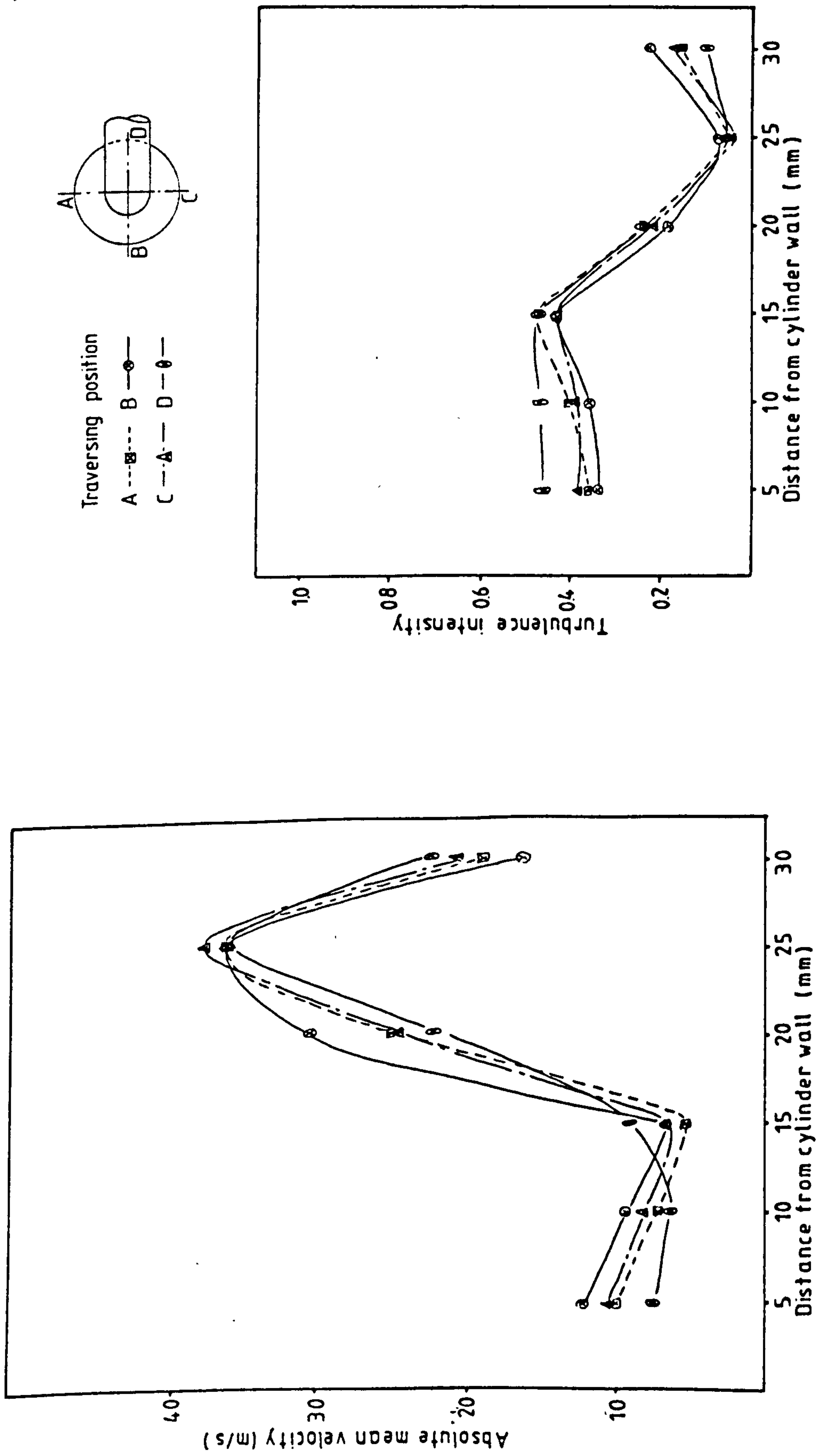


Fig. 5.35 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the variable ramp device at WOT. Wire orientation normal to (z, r) plane.

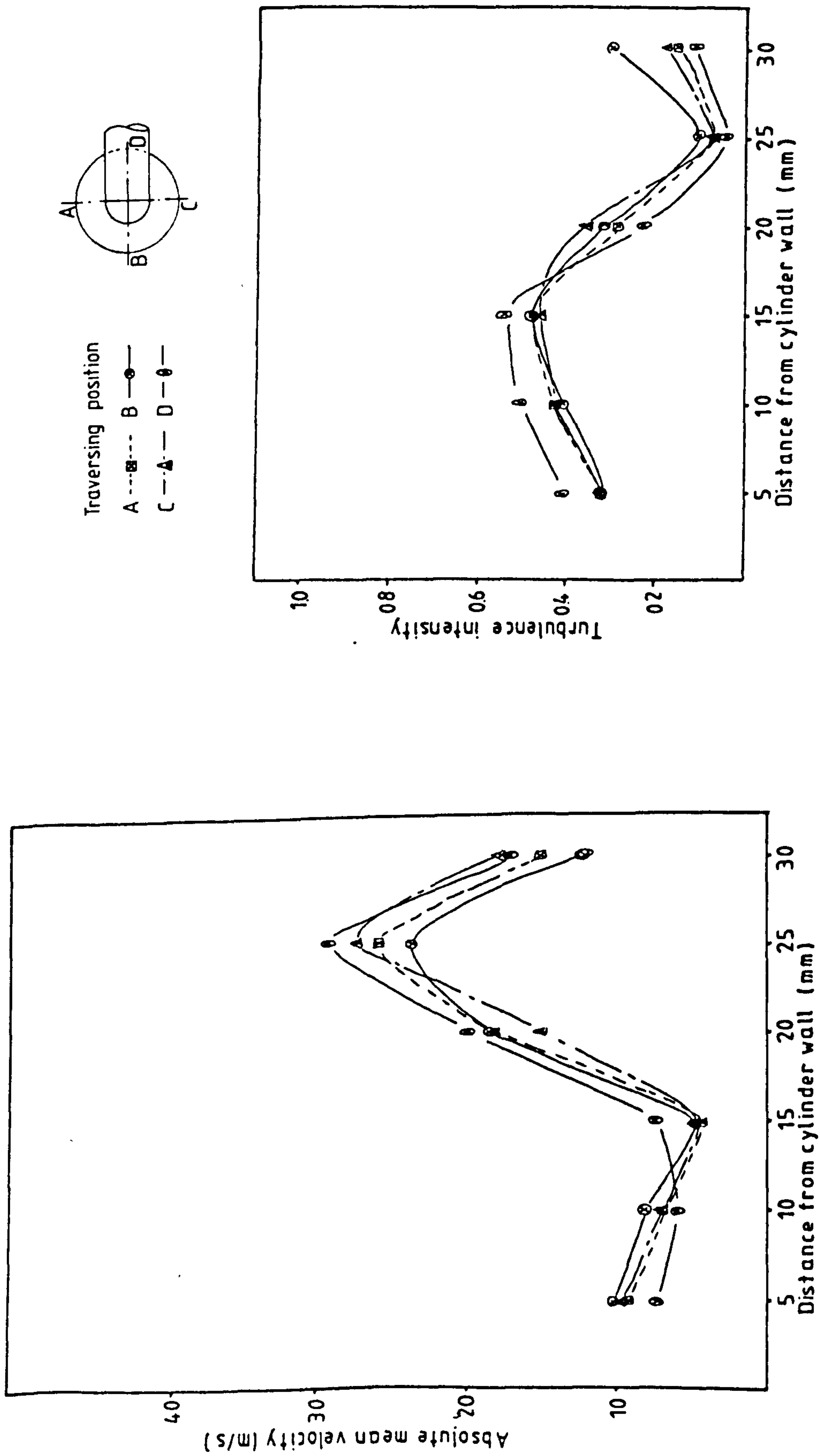


Fig. 5.36 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the variable ramp device at WOT. Wire orientation normal to (r, θ) plane.

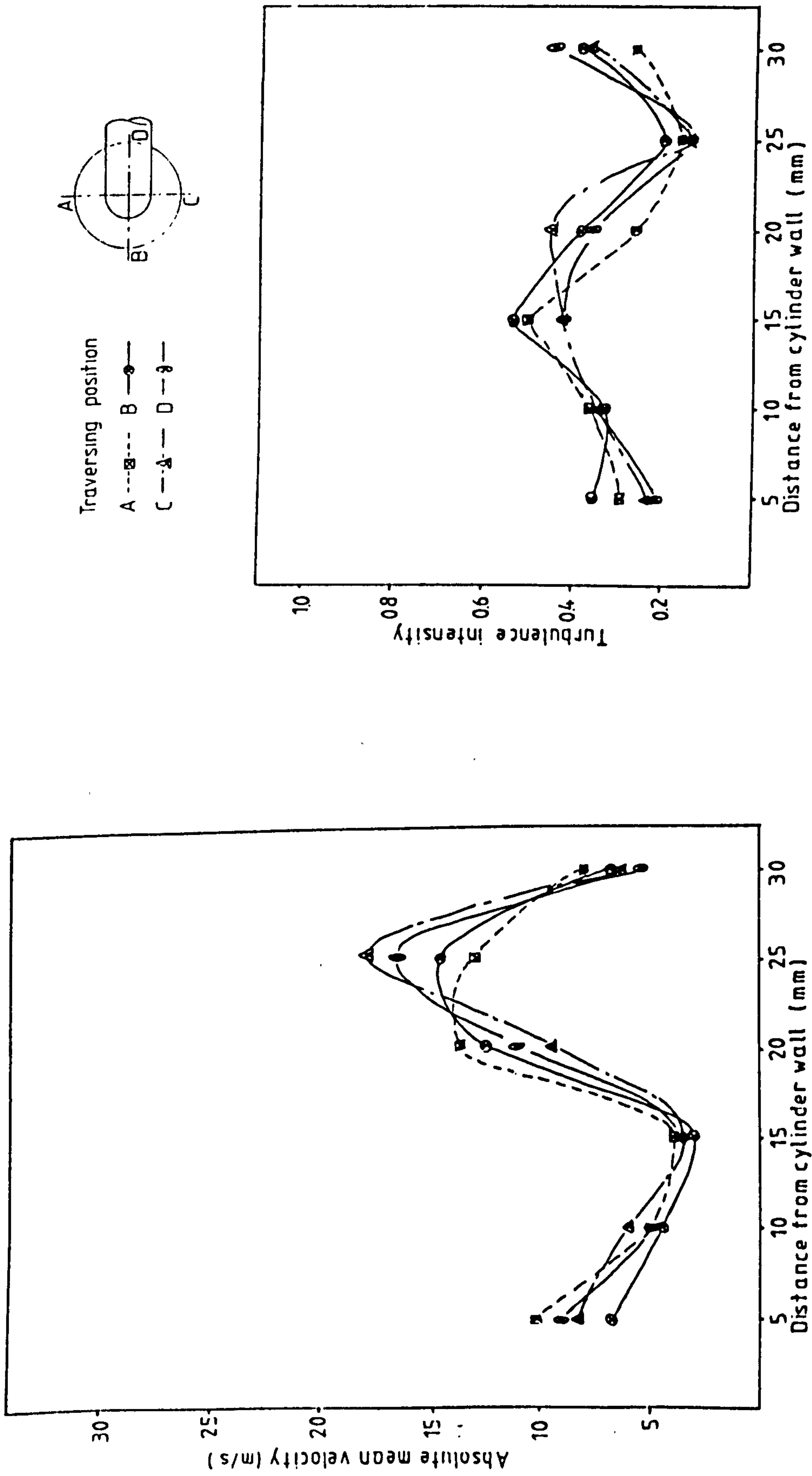


Fig. 5.37 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the butterfly valve device at $\beta = 30^\circ$. Wire orientation normal to (z, r) plane.

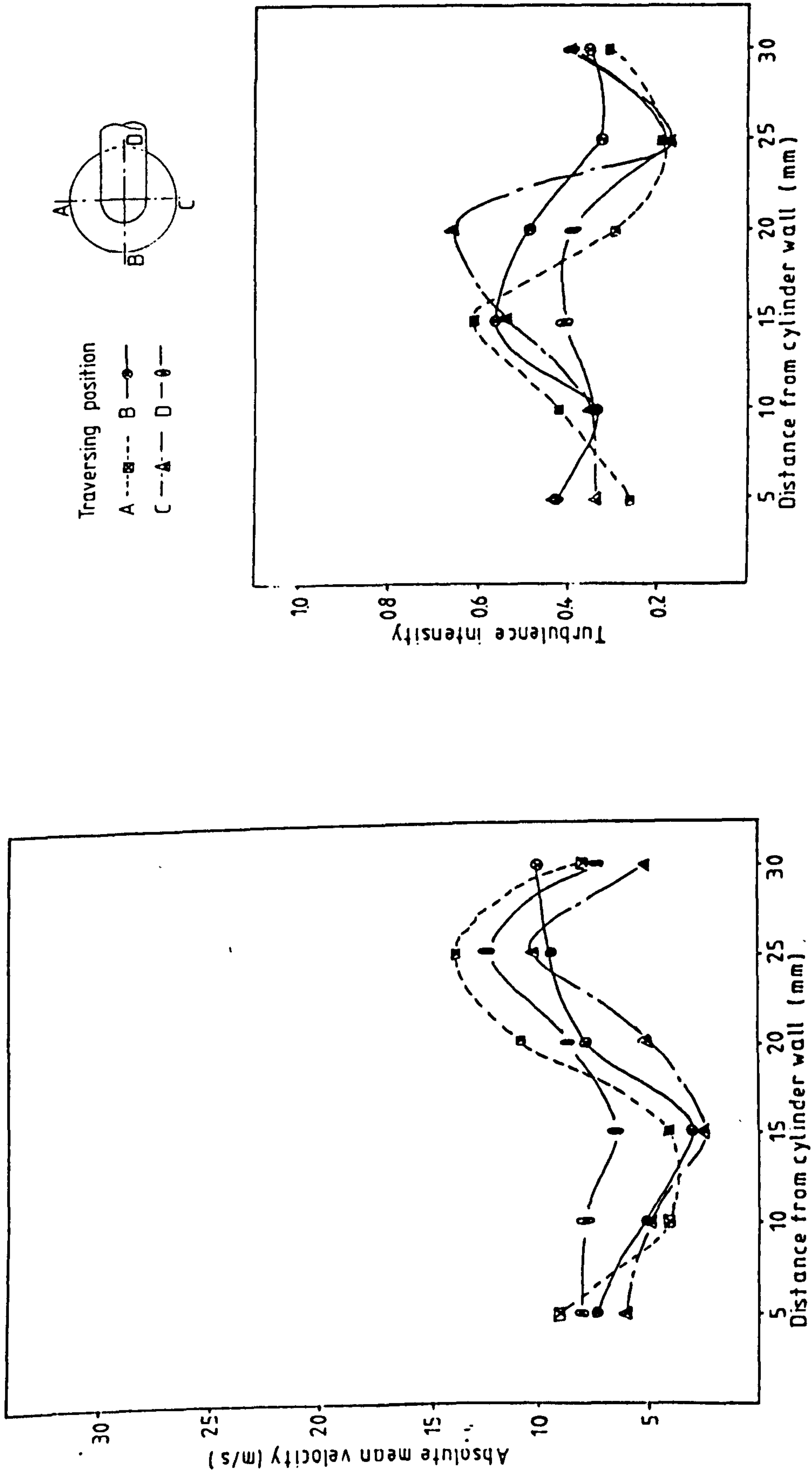


Fig. 5.38 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the butterfly valve device at $\beta = 30^\circ$. Wire orientation normal to (r, Θ) plane.

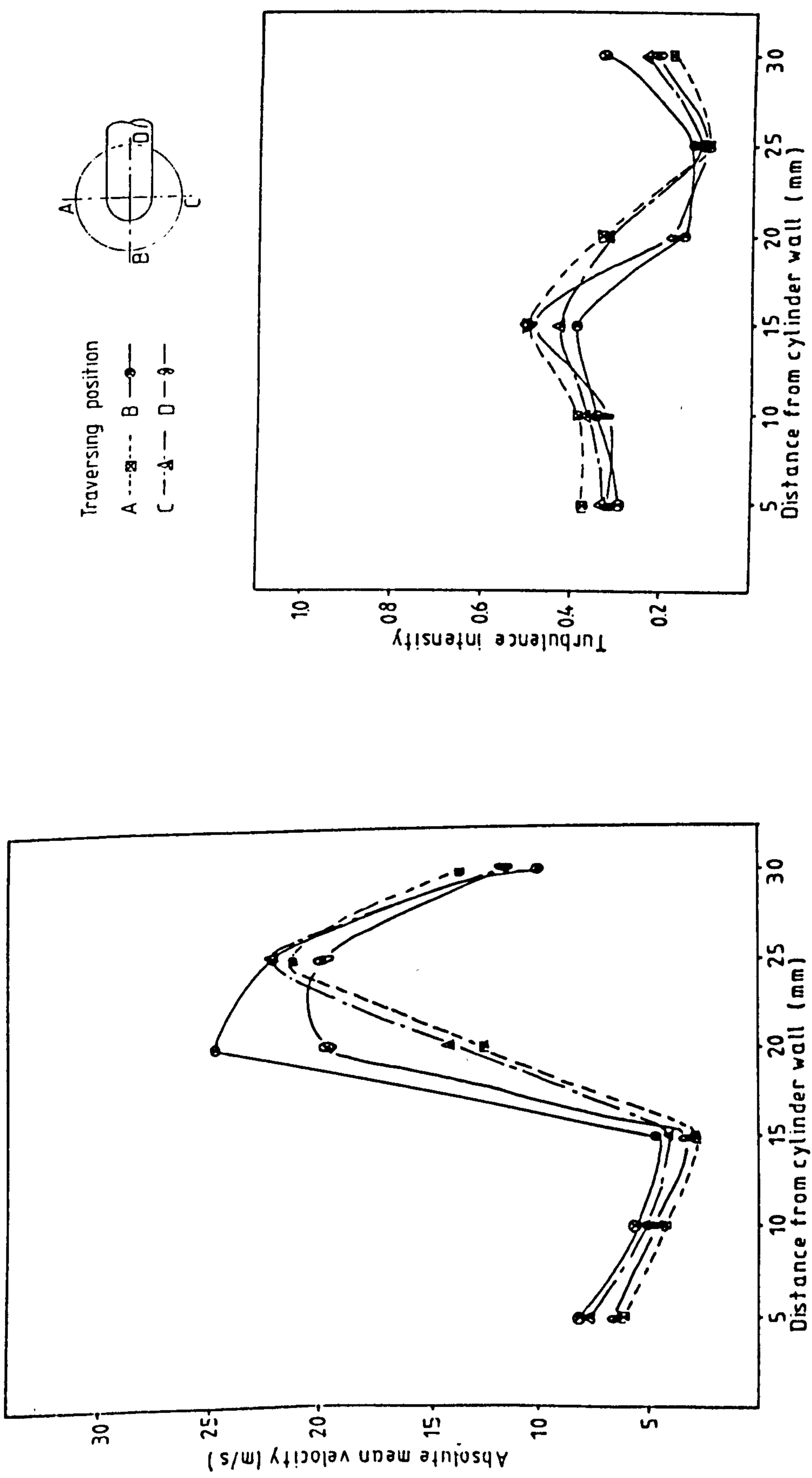


Fig. 5.39 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the butterfly valve device at $\beta = 50^\circ$. Wire orientation normal to (z, r) plane.

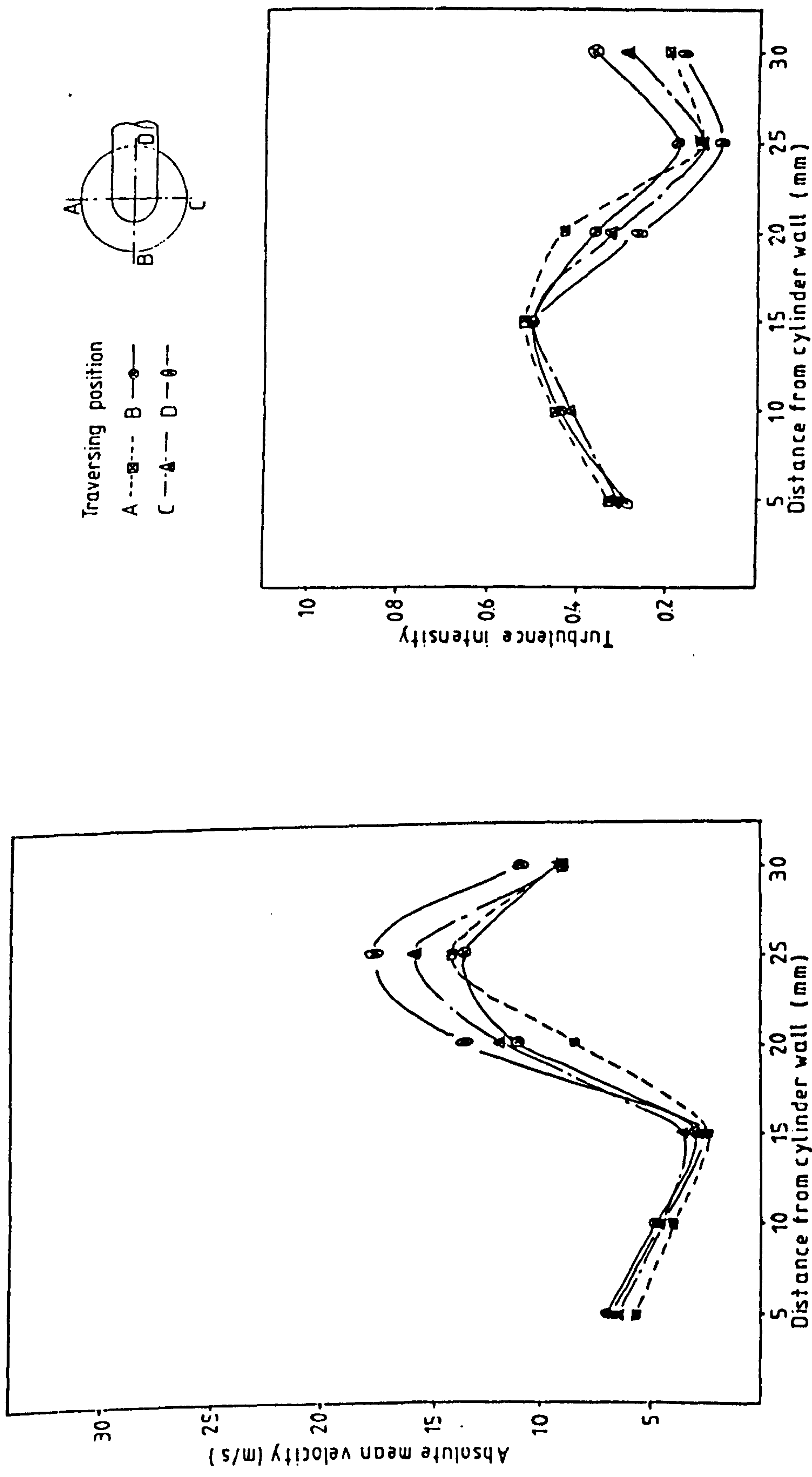


Fig. 5.40 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the butterfly valve device at $\beta = 50^\circ$. Wire orientation normal to (r, θ) plane.

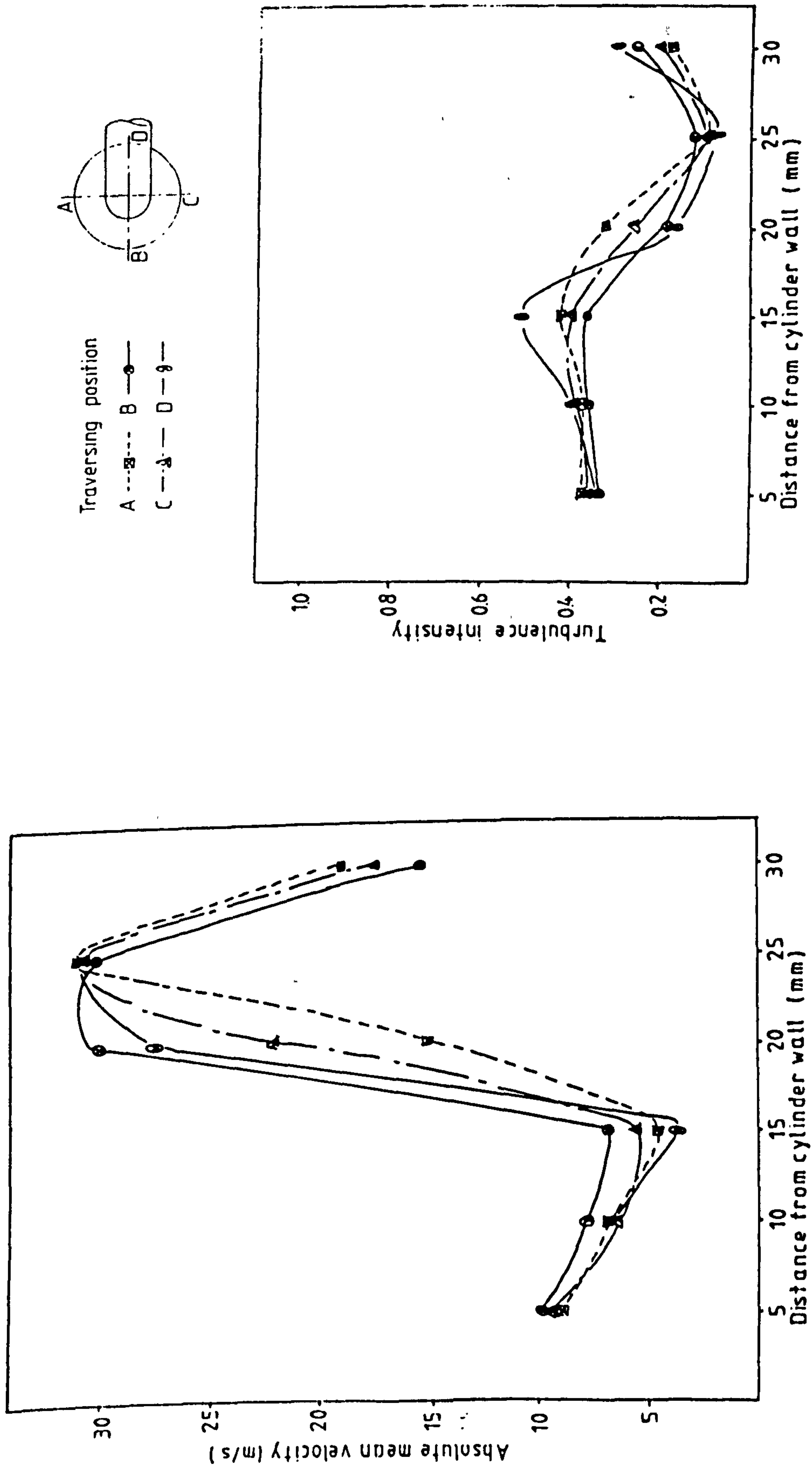


Fig. 5.41 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the butterfly valve device at $\beta = 65^\circ$. Wire orientation normal to (z, r) plane.

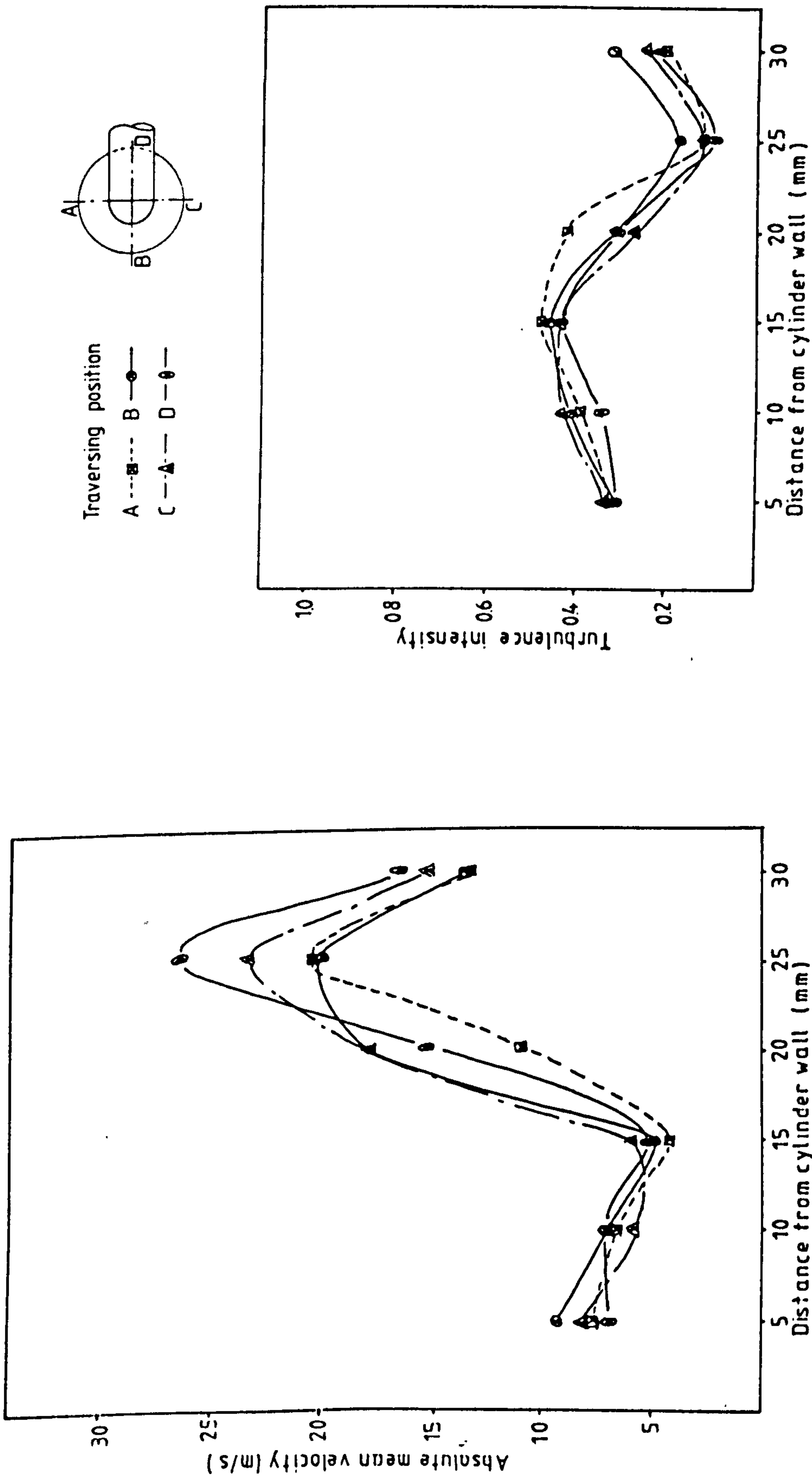


Fig. 5.42 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the butterfly valve device at $\beta = 65^\circ$. Wire orientation normal to (r, θ) plane.

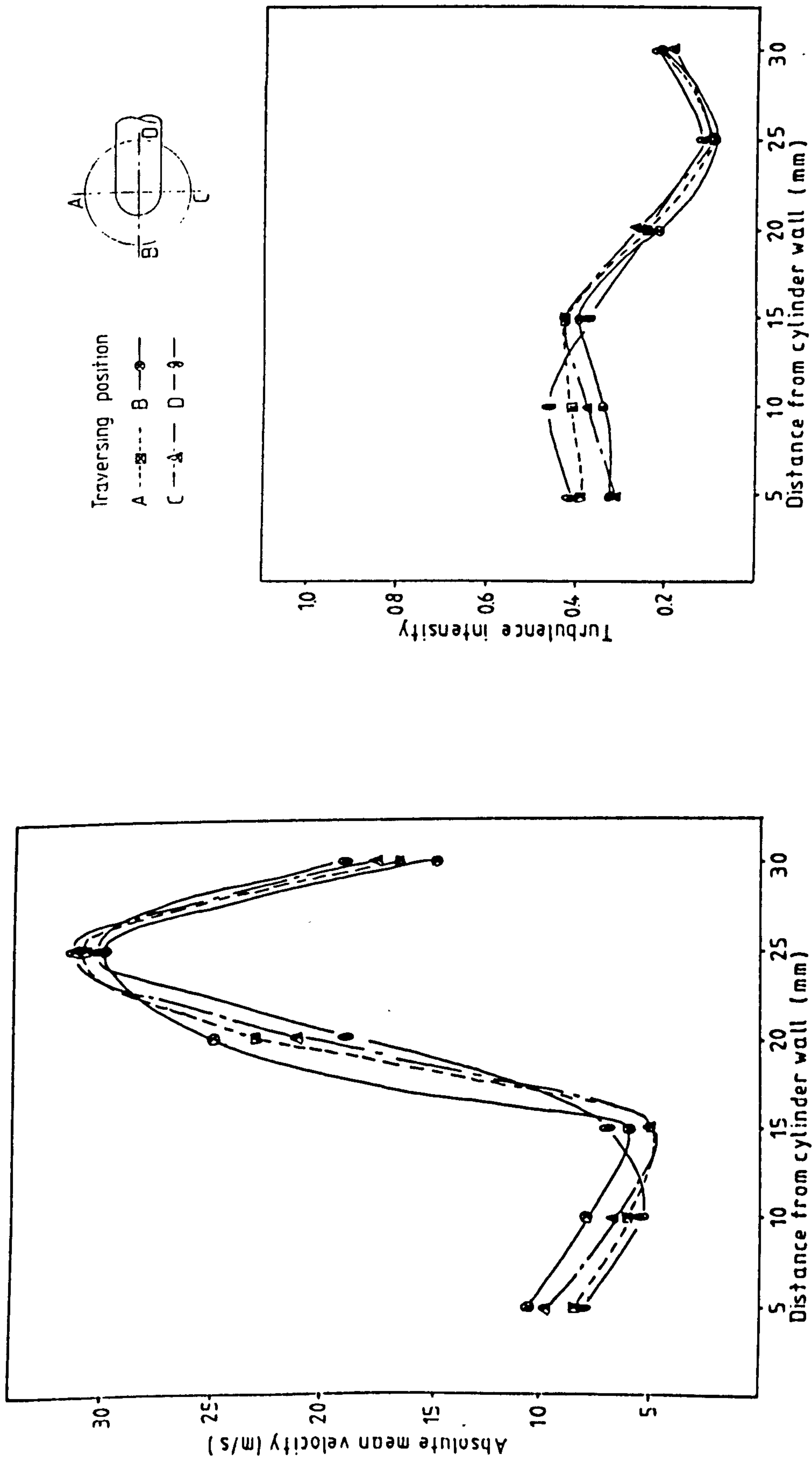


Fig. 5.43 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the butterfly valve device at $\beta = 75^\circ$. Wire orientation normal to (z, r) plane.

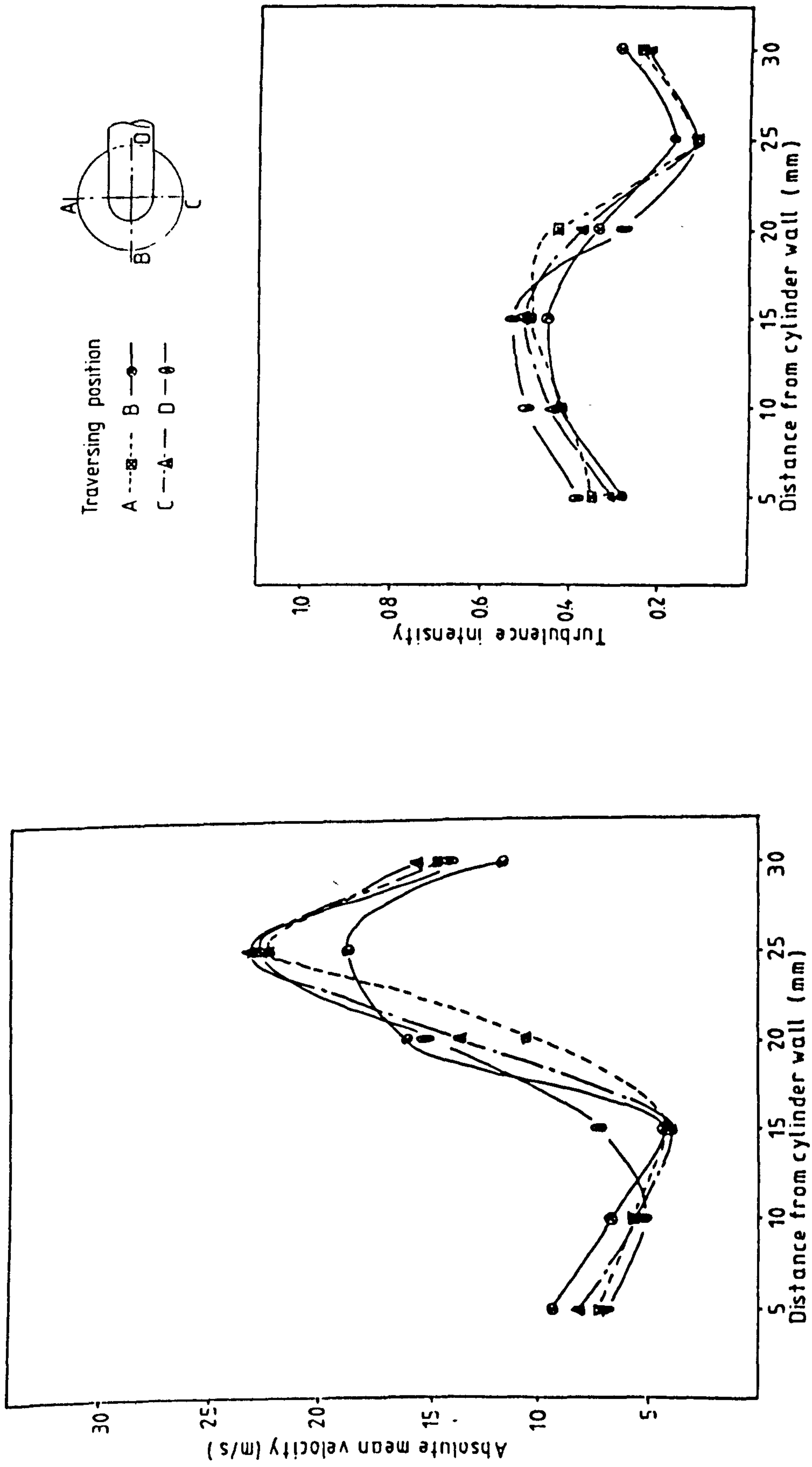


Fig. 5.44 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the butterfly valve device at $\beta = 30^\circ$. Wire orientation normal to (r, θ) plane.

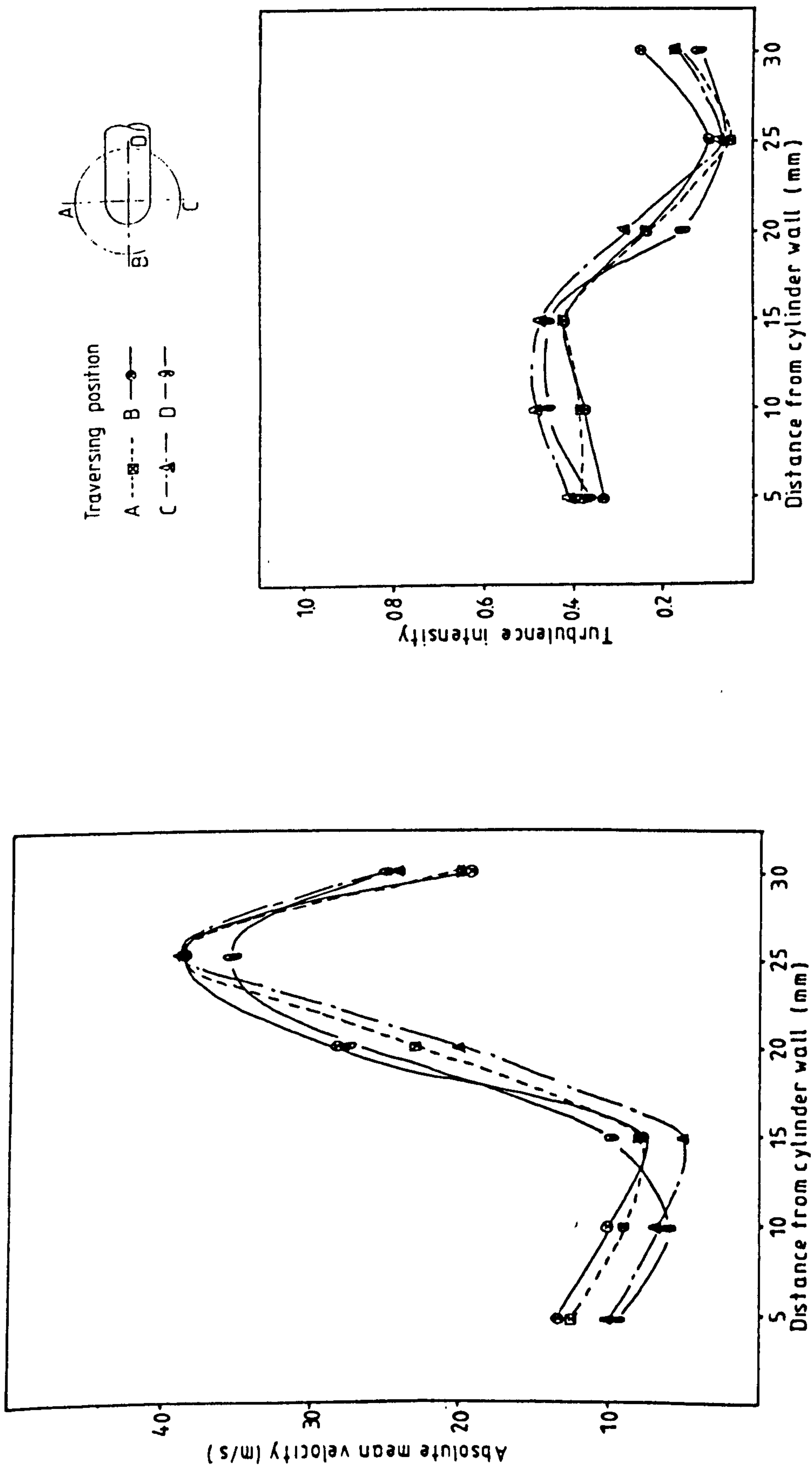


Fig. 5.45 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the butterfly valve device at WOT. Wire orientation normal to (z,r) plane.

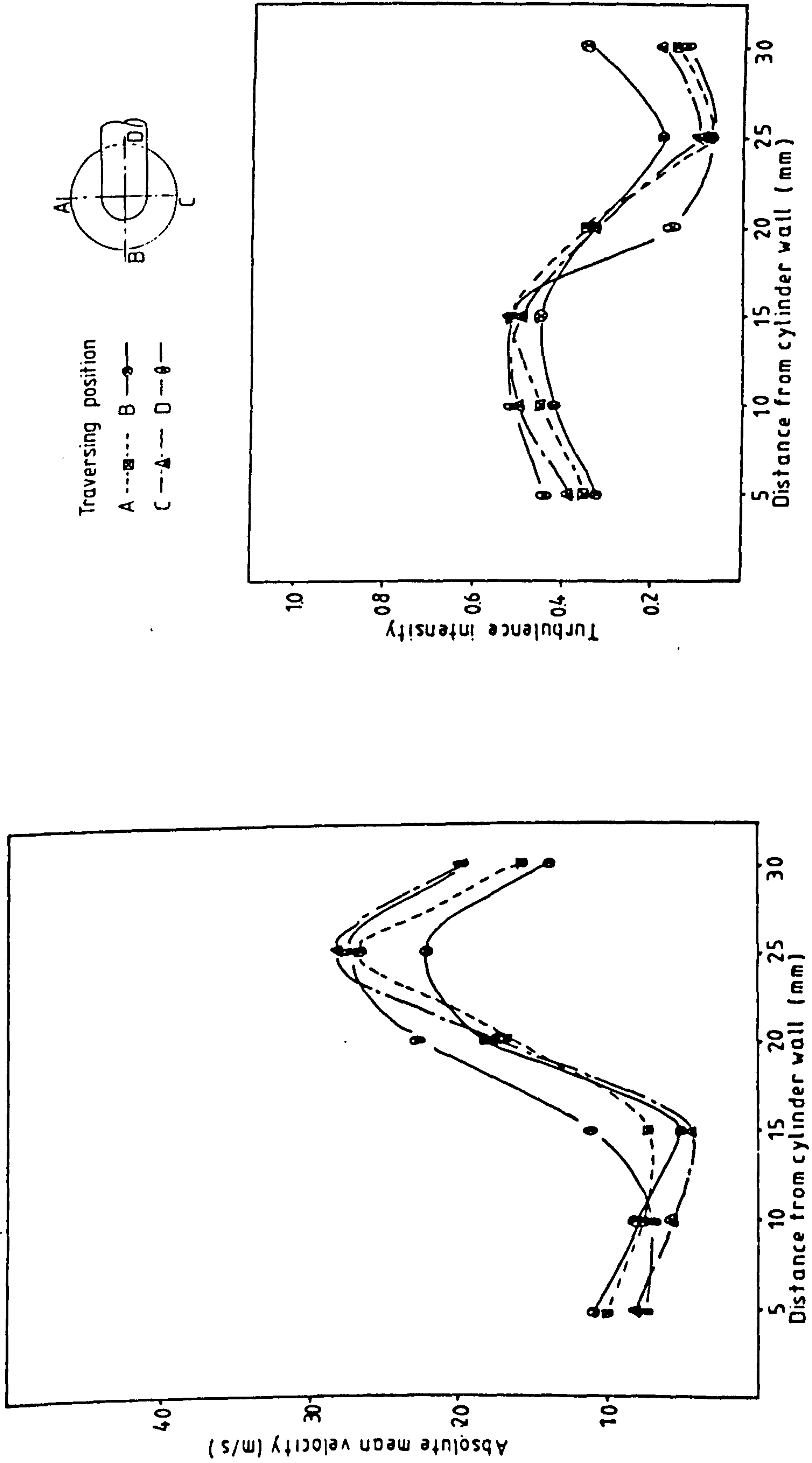


Fig. 5.46 Mean velocity (V) and turbulence intensity ($\sqrt{v^2} / V$) at Plane 3 for the butterfly valve device at WOT. Wire orientation normal to (r, θ) plane.

CHAPTER 6COMPUTER MODELS OF THE INDUCTION FLOW IN ALTERNATIVE THROTTLING SYSTEMS6.1 Introduction

The philosophy of the present research in relation to the induction flow and in-cylinder processes of the reciprocating engine was to pursue parallel programmes of flow visualization, flow measurement and computer modelling. The measurements which were described in Chapter 5 using hot-wire techniques, and the flow visualization pictures presented in Chapter 4 serve partly to provide new understanding of the effects of influential parameters such as geometry and boundary conditions, but also play the equally or perhaps more important role of validating the calculations.

The computer simulations are performed using computer methods to solve numerically a mathematical model, which comprises a set of differential equations, and which in turn describe the detailed flow processes as closely as possible. Although such methods can produce results more quickly and over a wider range of conditions than can experiments, they must embody approximations, geometrical, numerical and physical, which can give rise to appreciable errors in particular cases. Hence the need for experimental validation. A more detailed discussion of computational modelling and computer methods is presented in section 6.2.

The particular computer code used in the present study, a widely available package named PHOENICS, is a new generation computer code for simulating a wide range of fluid mechanics, heat and mass transfer, and combustion flows. This particular computer code was chosen primarily for its general-purpose nature, which in principle can be adopted to a variety of problems, and which can produce useful computations for diagnostic and design purposes both quickly and inexpensively. A more detailed description of the PHOENICS code structure and capability is presented in section 6.3.

Altogether sixteen test cases for which predictions have

been obtained with PHOENICS are presented and assessed in this chapter, eight of which concerned the simulation of three-dimensional steady flows in the two alternative throttling systems under different throttle settings, and a further eight simulating the corresponding in-cylinder processes. The unsteady cases will be dealt with in the following chapter.

The important dimensions and operating conditions, together with the geometric configurations of the induction flows, were primarily taken from the experimental models, and simulated as closely as possible. Compromises, however, had to be made. This was mainly due to the limitations of the calculation procedure in areas such as geometric configuration simulation and corresponding boundary conditions. These are explained and discussed in detail in sections 6.5 to 6.7, where the construction of individual models are presented.

The final sections of this chapter deal with the validation and comparison of the computed results with the corresponding experimental data, and the significant advantages of one system over the other and their consequent effects on engine performance.

6.2 Computational fluid mechanics

A theoretical prediction explores the consequences of a mathematical model rather than those of an actual physical representation. For the physical processes of present interest the mathematical model consists mainly of a set of coupled partial differential equations describing the conservation of mass, momentum and energy. If the analytical methods of mathematics were to be used for solving these equations, there would be little hope of predicting most phenomena of practical interest. A reference to any standard text on fluid mechanics leads to the conclusion that only a small fraction of the range of practical problems can be solved in closed form.

Fortunately, the development of numerical methods and the availability of large digital computers means that the implications of increasingly detailed mathematical models can be evaluated for many practical problems. The basis of the computational approach to problem solving is to replace the continuous information contained

in the exact solution of the differential equation with discrete values, and hence discretize the distribution of the problem's dependent variables. A grid can then be imagined to fill the domain of calculation and the values of variables sought at the grid points. [Algebraic equations for the variables in finite difference form can then be constructed and solved. The simplification inherent in the use of algebraic difference equations rather than simultaneous partial differential equations is the feature of numerical methods, which makes them so powerful and widely applicable.

6.2.1 Mathematical description of physical phenomena

The numerical solution of fluid flow and other related processes can be given when laws governing these processes have been expressed in mathematical form, generally in terms of differential equations.

The individual differential equations express a certain physical quantity as its dependent variable and implies that there must be a balance among the various factors that influence the variable. The dependent variables of these differential equations are usually specific properties, i.e. quantities expressed on a unit-mass basis.

The mathematical problem may be compactly expressed, with the aid of cartesian tensor notation, in terms of the following set of differential equations for viscous incompressible flow:

$$\rho \frac{\partial u_i}{\partial x_j} = 0 \quad (6.1)$$

$$\rho \left(\frac{\partial u_i}{\partial t} + \frac{\partial u_j u_i}{\partial x_j} \right) = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu_{eff} \frac{\partial u_i}{\partial x_j} \right) + S_i \quad (6.2)$$

$$\rho \left(\frac{\partial \phi}{\partial t} + \frac{\partial u_j \phi}{\partial x_j} \right) = \frac{\partial}{\partial x_j} \left(\Gamma_{eff} \frac{\partial \phi}{\partial x_j} \right) + S_\phi \quad (6.3)$$

which express the laws of conservation of mass, momentum and scalar property ϕ , respectively. Here the dependent variables are the time average values of the velocities u_i , the pressure P , and ϕ , which stands for such scalar quantities as enthalpy, concentration, kinetic energy or dissipation rate of turbulence, etc. The symbols S_i and S_ϕ stand for additional sources associated with such phenomena as natural convection, chemical reaction, etc., while ρ , μ_{eff} and $\Gamma_{\phi eff}$ are respectively the density, viscosity and exchange coefficient for ϕ . The subscript "eff" appended to the latter two indicates that, for turbulent flows, they are sometimes ascribed "effective" values, deduced from turbulence quantities such as kinetic energy and dissipation rate of turbulence. This will be discussed in detail later on in section 6.4.

Not all diffusion flows, however, are governed by the gradient of relevant variable. But the use of

$$\frac{\partial}{\partial \kappa_j} \left(\Gamma_{\phi eff} \frac{\partial \phi}{\partial \kappa_j} \right) \quad (6.4)$$

in equation 6.3 does not limit the general equation to gradient-driven diffusion processes. Whatever can not be fitted into the nominal diffusion term can always be expressed as a part of the source term. This concept of general equation enables the formulation of a general numerical method, and enables the preparation of general-purpose computer programs such as PHOENICS, which will be discussed in the following section in detail.

6.2.2 Nature of numerical methods

The numerical solution of a differential equation consists of numbers from which the distribution of the dependent variable ϕ can be constructed. The numerical analysis, however, is concerned with only a finite number of numerical values as the outcome, although this number can and must be made large enough for practical purposes. Most numerical methods are based on employment of the values of ϕ at a number of given points as the primary unknowns.

Thus the numerical method treats as its basic unknowns the values of the dependent variable at a finite number of locations, called the grid points, in the calculation domain. [The method has to include the tasks of providing a set of algebraic equations for these unknowns and of prescribing an algorithm for solving the equations. By focusing attention on the values at the grid points, the continuous information contained in the exact solution of the differential equation is replaced with discrete values. It is this systematic discretization of space and of the dependent variable that makes it possible to replace the governing differential equations with simple algebraic equations, which can be solved with relative ease. There are two alternative versions of the discretization method, which are finite-difference and finite-element. The distinction between the two methods result from the ways of choosing the profiles and deriving the discretization equations. In the finite-element method and in most weighted-residual methods, the assumed variation of ϕ consisting of the grid-point values and the interpolation functions or profiles between the grid points is taken as the approximate solution. In the finite-difference method, however, only the grid-point values of ϕ are considered to constitute the solution, without any explicit reference as to how ϕ varies between the grid points. In PHOENICS the control-volume approach adopts a view similar to that of the finite-difference method. This will be discussed in more detail in the following section.

6.3 PHOENICS computer code

PHOENICS is a new generation computer code for simulating a wide range of phenomena in fluid mechanics, heat and mass transfer, combustion and other chemical systems. These flows may involve one, two or three dimensions, one, two or three interpenetrating phases, fixed or moving boundaries, laminar or turbulent flow.

In view of the expense and time involved in the development of computer codes, there has always been a need for a widely available, readily adaptable, general-purpose computer code which can be adapted

quickly to a novel problem and which can produce useful engineering computations quickly and inexpensively. PHOENICS has been constructed by CHAM with these needs in mind (65).

PHOENICS is an acronym, standing for Parabolic, Hyperbolic or Elliptic Numerical Integration Code Series. Built into PHOENICS are the major 'balance' equations of fluid physics, applied to a large number of sub-domains into which the field of study is artificially divided. When supplied with appropriate information concerning the physical properties of the participating media, the geometrical and other constraints and boundary conditions, it computes the corresponding solutions to the relevant system of coupled partial differential equations in finite difference form.

6.3.1 The structural principle of PHOENICS

The central core of PHOENICS is not transparent to the user. It is embodied in a group of sub-routines called EARTH to which the user does not have access. EARTH is thus the same for all users and all uses. On the other hand, users have access to a variety of auxiliary programs called SATELLITES and to additional sub-routines called GROUND stations.

EARTH sets up the sub-domain grid, allocates the computer storage, sets up the balance equations for each variable, obtains the solutions to these equations in an orderly and convergent fashion and arranges for the printing of required output. It is this section of the computer code which can in principle handle steady, unsteady, one, two or three space dimensions, effects of compressibility, chemical reaction and moving boundaries, and both empty domains and those partially filled by solid structures.

The SATELLITES of PHOENICS system are also separate codes. Their function is to provide EARTH with, among other things, the following data:

- the nature of the process to be simulated,
- the properties of the materials in question,
- the shape and size of domains of interest,
- the grid mesh to be employed.

The SATELLITE codes are much smaller than EARTH and consist,

for the most part, of a main program which supplies a data file, and auxiliary sub-routines which supply information about geometry, initial fields and other special data (i.e. see Appendix A).

Whereas SATELLITES deliver problem-defining data for EARTH to process but have finished their work before the processing begins, GROUND station sub-routines participate in this processing. A GROUND station, like the SATELLITE which has supplied the data, contains special features appropriate to the problem in question. Its sub-routines are accessible to users, and they can modify any of the data which the SATELLITE has provided. The communication with EARTH is performed by way of service sub-routines. For more information on GROUND station and its service sub-routines reference should be made to Appendix A.

6.3.2 Mathematical basis of PHOENICS

PHOENICS is equipped to solve for 25 named variables, and as many more as the user cares to specify. These include variables such as the following, appropriate to the present application: pressure correction, velocities in three directions, namely x, y and z, turbulent kinetic energy and its rate of dissipation.

The independent variables in PHOENICS are the quantities x, y, z, together with time, t. In rectangular cartesian coordinates, x, y and z are the distances in the three orthogonal directions. In the cylindrical polar coordinate option, x is the azimuthal angle and y and z the distances, respectively, normal and parallel to the axis of symmetry.

All the partial differential equations, of which PHOENICS solves a discretized version, are expressed in a single form, namely:

$$\frac{\partial}{\partial t} (r \rho \phi) - \text{div} (r \rho \vec{u} \phi - r \Gamma_{\phi} \text{grad} \phi) = r S_{\phi} \quad (6.5)$$

where ϕ can stand for the dependent variables, and r , ρ , $\vec{u}$, Γ_{ϕ} and S_{ϕ} represent, for the phase appropriate to ϕ , respectively the volume fraction, density, velocity vector, exchange coefficient and source term.

PHOENICS embodies a 'finite-volume' or 'finite-domain' (FDE) formulation of the differential equations describing property conservation. They are derived by integration of the differential equations over control volumes of finite size (cells), which, taken together, wholly fill the computational domain.

The cells have six sides and eight corners, in the general three-dimensional case. Within each cell is a typical point called a grid node, for which the fluid properties are regarded as representative of the whole cell. Cells and nodes for velocity components are, however, 'staggered' relative to those of all other variables.

The integration entails interpolation assumptions concerning values of ϕ and of ϕ -gradients, prevailing at cell boundaries. Integration leads to finite-domain equations having the algebraic form:

$$a_p \phi_p = a_N \phi_N + a_S \phi_S + a_E \phi_E + a_W \phi_W + a_H \phi_H + a_L \phi_L + a_T \phi_T + S \quad (6.6)$$

where a_p , a_N , a_S , etc. are coefficient, subscript P denotes grid points, subscripts N, S, E, W, H, L denote neighbouring nodes and T the grid node at earlier time. S is a representation of the source appropriate to ϕ for the cell. In equation 6.5 the a's, in effect, express the influences of convection and diffusion processes across the cell boundaries.

As mentioned in the previous section, the finite domain equations in PHOENICS differ from finite-difference and finite-element equations, in as much as no Taylor-series expansion is used. Hence, the values of coefficients differ from those for finite-difference equations, and neither variational principle nor Galerkin weighting is used, hence the number of nodes referred to (8 for three-dimensional transient problem) differ from finite-element equations, which is usually more.

6.3.3 The solution procedure

The finite-domain equations are solved in PHOENICS by a variant of the SIMPLE (67) procedure. SIMPLE stands for Semi-Implicit Method for Pressure-Linked Equations. However, "semi" is inappropriate since the method is fully implicit.

The solution procedure assumes the main dependent variables to be velocities and pressure, which are computed on a number of staggered, interlacing grids, each of which is associated with a particular variable. Typically, the hybrid central upwind difference scheme is employed and the solution algorithms are sufficiently implicit to obviate the need to approach the steady state via the time evolution of the flow, as is required by wholly explicit methods. This method has the advantage of reducing the needlessly expensive computation required by methods attempting to follow the time evolution of the flow in arriving at the steady state solution, especially when an explicit formulation is employed. Furthermore, because this procedure computes the variable fields successively, rather than simultaneously, it is highly flexible in respect of the methods of solution (i.e. jacobian point-by-point, slab-by-slab, whole-field, etc.), which it will admit for the difference equations.

The essential ideas can be summarised as follows:

- The pressure field is first guessed.
- The corresponding velocity fields are then computed.
- The resulting errors in the continuity equation are computed.
- These errors are used as quantitative indicators of corrections which must be made to the pressure, and of the corresponding velocity corrections.

The actual values of the pressure correction are obtained by the solution of a set of simultaneous finite-volume equations.

The order of visitation which is selected when variables at cell-centres are up-dated has a significant influence on the ability of the code to handle fine-grid problems. The complete set of cells is regarded as consisting of one-cell-thick slabs, extending in the x and y directions, and piled one on top of the other in the z direction.

A single sweep therefore starts with attention focussed on the bottom (low- z) slab of cells. The equations are solved for all the cells in this slab, the values of ϕ at the next higher slab being regarded as known. Attention then passes to the second slab, the ϕ -values there being adjusted by reference to those in the slabs both above and below. Then the next higher slab is considered and so on until the adjustment sweep has been completed. At the end of

a sweep (unless the flow is parabolic) the process must be repeated until the prescribed level of round-off error is attained.

When the flow is transient, the same iterative sweep procedure is employed. However, the number of sweeps needed to achieve convergence at a particular time step reduces greatly with the magnitude of the time interval.

6.3.4 Porosity specification

Porosities are used to modify the regular cartesian or polar grid cells. There are four porosities for each cell: the cell or volume porosity represents the proportion of the cell volume available for occupancy by the fluid, and the other three, the East, North and High porosities represent the proportions of the large-x, large-y and large-z (i.e. East, North and High) faces of the cell which are available for flow.

The area A entering diffusion and convection terms, and volume V which enters time-dependent and source terms, may in PHOENICS differ from the products of cell-side lengths which represent the normal cell-face areas and volumes.

Specially:

$$A = A_{\text{nom}} \cdot f_A \quad (6.7)$$

and

$$V = V_{\text{nom}} \cdot f_V \quad (6.8)$$

where A_{nom} and V_{nom} stand for nominal values and f_A and f_V are the porosity factors, to which values other than unity can be ascribed.

Porosity values of unity correspond to the situation of no blockage. Values of zero correspond to completely-blocked cells or faces. Completely-blocked cells are effectively excluded from the calculation domain, though the program goes through the motions of solving equations in such cells. Values of porosity between zero and unity represent partial blockage.

Once the distributions of the four types of porosity factors (North-area, East-area, High-area and volume) are specified, the

built-in logic of PHOENICS ensures their effects on diffusion fluxes, etc. are properly accounted for. However, if an internal obstruction does more than restrict the access of the fluid, for instance by exerting a momentum sink or provide a heat source, additional information about these sinks or sources have to be provided, for more than changes of areas and volumes are involved.

While it is true that with the aid of full and partial blockage of cells and cell-faces complicated geometries can be constructed, the inability to prescribe some types of boundary conditions, such as wall friction on partially blocked cells, is the major drawback in using porosity as a means of constructing geometries such as sloping solid walls. In such cases, use of non-orthogonal body fitted coordinates to deform the calculation grid would be much more appropriate. In the present studies, however, this option was not available, and porosities had to be used for the construction of the internal flow geometries, as will be seen later on in this chapter.

Full details of the concept of cell porosities used in PHOENICS are found in Ref. 66.

6.3.5 Incorporation of boundary conditions

The boundary information required by the finite-domain equations effectively takes the form of relations connecting the boundary fluxes and values of the entity in question to its values at neighbouring grid nodes.

All boundary conditions in PHOENICS are inserted as sources or sinks of one or more variables. These are represented by linear expressions of the form:

$$\text{SOURCE} = C_{\phi} (V_{\phi} - \phi) \quad (6.9)$$

where C_{ϕ} and V_{ϕ} are the "coefficient" and "value" which are prescribed separately for each variable for each region and ϕ is the calculated local grid-node value of the variable in question.

When boundary conditions involve mass inflow or outflow, the above source expression becomes:

$$\text{Source of Mass, } \dot{m} = C_{\text{mass}} (V_{\text{mass}} - P) \quad (6.10)$$

Thus, the value V_{mass} now has the physical significance of an external pressure, and the coefficient, C_{mass} , that of the reciprocal flow resistance between the internal grid-node, where the pressure is P and external pressure V_{mass} .

In the presence of mass inflow, the source-term expression for all other variables becomes:

$$\text{Source of } \phi = (C_{\phi} + \left[\dot{m} \right]) * (V_{\phi} - \phi) \quad (6.11)$$

The value V_{ϕ} now represents the value of ϕ in the incoming stream. The $[]$ symbolizes that only inflow is of influence. In circumstances where convective transport across the boundary is dominant C_{ϕ} would be set to zero. This has usually been the case in the models tested here and will be discussed later.

The sources or sinks may also be specified as totals for each cell, or per unit volume, or per unit area, for any of the six faces of the cell. The latter has been the case for all the boundary condition specified in this chapter.

The specification of boundary conditions for individual models tested will be discussed later in the appropriate sections.

6.4 Mathematical models of turbulence

In principle, there is no need to adopt special practices for turbulent flow since the Navier-stoke equations are exact in these terms and apply as readily to a turbulent motion as to a laminar one. This, however, is not a realisable route at present, for many important details of turbulence are small-scale in character. To solve the equations we must, therefore, use a numerical procedure that calculates the values of all variables at a large number of discrete points in space and time, which would far exceed the storage capacity of existing computers.

Fortunately in most engineering applications there is concern only with time-averaged effects of turbulence, and since these vary

more gradually in space excessively fine grids are not needed.

There are two main classes of turbulence model, namely statistical flux models (SFM) and large-eddy simulation (LES).

In the SFM the basic conservation equations governing the instantaneous flow are ensemble-averaged to yield a new set containing additional unknowns which have the significance of turbulent fluxes of the entity in question (e.g. the "Reynolds stress" in the case of momentum equations). Closure of the equation set is then effected by deriving additional equations, of either algebraic or differential form, for the new unknowns, in which process certain approximations and assumptions necessarily enter which render the result inexact. The success or failure of such methods therefore resides in the validity of the approximations which they embody.

In the LES the averaging process is limited to eddies below a certain scale which can hopefully be made of the same order as the mesh size of the computational grids for which numerical calculations are feasible. Motions larger than this are calculated in the normal way, i.e. as for laminar flow. The new unknowns arising from the averaging process (termed "subgrid-scale Reynolds stresses" in the case of the momentum equations) are approximated by a SFM, but the small-scale turbulence is believed to possess certain regularities (e.g. isotropy) which should allow them to be modelled accurately in particularly simple ways.

A more detailed exposition of these methods can be found in Refs. 68 and 69, among others.

Of the two approaches the LES appears to be more direct and potentially more accurate, since it only approximates that part of the eddy-scale spectrum which is not accessible to direct calculation. Nevertheless, in common with other CFD codes, PHOENICS adopts the SFM approach. LES is less well-developed and remains computationally expensive. It does not, for example, allow the full economies of the restriction to axisymmetric flow to be realized for the simple reason that even in such flows the turbulence structure remains three-dimensional and must be treated as such in the calculations.

6.4.1 Turbulent viscosity or eddy viscosity models

The first move towards a model of turbulence can be attributed to Boussinesq (70), who more than one hundred years ago suggested that the effective turbulent shear stress arising from the cross-correlation of fluctuating velocities could be replaced by the product of mean velocity gradient and a quantity termed the "turbulent viscosity". The introduction of turbulent viscosity provided a framework for the construction of turbulence models.

There are two approaches for the turbulent viscosity modelling. The first approach, which is known as the Mixing Length Hypothesis (MLH), was invented by Prandtl (71). MLH is a simple approach in as much as it requires no additional differential equations to be solved, and provided that good choices are made for the mixing-length distributions, it allows realistic predictions to be made of the velocity and shear-stress distributions, and the general behaviour, of boundary-layer flows. On the other hand, MLH takes no account of processes of convection or diffusion of turbulence, and it has shown itself to be incapable of predicting recirculating flows.

The second approach, which is used in PHOENICS for the present study, overcomes the shortcomings of MLH by using a turbulence property (i.e. say square root of time-averaged turbulence kinetic energy), instead of relating the random velocity of turbulence to a mean velocity gradient. The differential transport equations for turbulence properties enables account to be taken of convective and diffusive influences of neighbouring regions on the local turbulence energy. Among such models are: the one-differential equation models of Prandtl (72), Bradshaw (73), Nee and Kovaszny (74), the two-differential equation models of Kolmogorov (75), Spalding (76), and Jones and Launder (77).

One of the simplest kinds of model that permits prediction of both near-wall and free-shear flow phenomena without adjustments to constants or functions is the two-differential equation $K-\epsilon$ turbulence model, Jones and Launder (77). It successfully accounts for many low Reynolds-number features of turbulence, and its use has led to accurate predictions of flows with recirculation as well as those of boundary-layer kind.

The major drawback with $K-\epsilon$ turbulence model, however, is its isotropic viscosity formulation, which renders it incapable of

fully connecting the stress and strain fields in turbulent flows with more than one significant shear-stress component.

Alternatively, algebraic and Reynold stress models have been applied with certain amount of success to anisotropic flows (Hanjalić and Launder (78)). In most cases, however, these algebraic-stress formulae can give rise to very complicated non-linear equations for stress components and, for recirculating flows, may seriously complicate the task of solution.

While it might be true that the present study could have benefited from such alternative turbulence models (i.e. anisotropic nature of flow in some regions of the throttling system and cylinder), the complications involved in such formulations resulted in the choice of the more commonly used and readily available (i.e. in PHOENICS) two-equation $K-\epsilon$ model.

6.4.2 The two-equation ($K-\epsilon$) model of turbulence

The approach to the turbulence models commonly used for recirculating flows, and indeed used in the present study, is a form of two-equation ($K-\epsilon$) model first introduced by Jones and Launder, 1972 (77). This involves an assumed linear relationship between the Reynolds stress and rate of strain:

$$-\rho \overline{u_i u_j} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \rho \delta_{ij} K \quad (6.12)$$

The turbulent viscosity or eddy viscosity is then given by:

$$\mu_t = C_\mu \rho K^2 / \epsilon \quad (6.13)$$

where $K \left(\equiv \frac{\overline{u_i u_i}}{2} \right)$ and ϵ are turbulence kinetic energy and its dissipation rate, respectively, the values of which are obtained from the solution of the transport equations for K and ϵ , which may be expressed as following (79):

$$\frac{DK}{Dt} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left(\frac{\mu_t}{\rho} \frac{\partial K}{\partial x_j} \right) + \frac{\mu_t}{\rho} \frac{\partial u_i}{\partial x_j} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \epsilon \quad (6.14)$$

and

$$\frac{D\varepsilon}{Dt} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left(\frac{\mu_t}{\delta_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{C_1 \mu_t}{\rho} \frac{\varepsilon}{K} \frac{\partial u_i}{\partial x_j} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + C_2 \frac{\varepsilon^2}{K} \quad (6.15)$$

The constants appearing in equations 6.14 and 6.15 having the values typically:

$$C_\mu = 0.09; \quad C_1 = 1.44; \quad C_2 = 1.92; \quad \delta_k = 1.0; \quad \delta_\varepsilon = 1.3;$$

The effective viscosity is then calculated from:

$$\mu_{eff} = C_\mu \rho K^2 / \varepsilon + \mu \quad (6.16)$$

The form of the model which has been presented above is valid for fully-developed turbulent flows. Close to solid walls, there are inevitably regions where the local Reynolds number of turbulence is so small that viscous effects predominate over turbulent ones. In such regions wall-function methods are used which will be discussed in the following sections.

6.5 Steady-flow simulation for the variable ramp throttling system

The four cases of the variable ramp throttling system for which predictions were obtained with PHOENICS are presented in this section. They were modelled in such a way as to simulate the geometries and flow conditions as closely as possible to those encountered in the flow measurement studies, namely for ramp angles 110°, 130°, 150° and 180°.

The following sub-sections contain descriptions which include the operating conditions, boundary conditions, number of grid nodes used and geometric configuration.

6.5.1 Grid arrangement and geometric configuration

The major problem concerning the prescription and simulation of the geometry for the flow domain arises from the highly irregular geometry of the system under study. The flow from the circular inlet pipe expands into the rectangular throttling and ramp section, and then leaves the system through a circular pipe.

When the simulations were undertaken there were two geometrical options for the finite domain grid, namely polar and cartesian. For the variable ramp cases the cartesian coordinate option was used because the geometry of the system appeared to fit a cartesian coordinate more accurately.

The overall geometry of the flow domain was treated as consisting of two independent parts:

- 1) The inlet and outlet pipe, plus the rectangular throttling section, which is common for all the test cases and is independent of the throttle setting.

- 2) The variable ramp configuration which is different for each test case and depends on the throttle setting.

The number of grid nodes used for the variable ramp models was 3654 (i.e. $NX=7$, $NY=18$, $NZ=29$), with z being the axial direction of the flow. The grid was not, however, distributed evenly in any given direction. A finer grid distribution was adopted for regions of particular interest, such as the throttling and ramp regions, where finer details of flow were to be studied. A certain amount of grid optimization work was carried out during the course of the study, which is discussed later towards the end of this section.

The simulation of the circular inlet and outlet pipe geometries, together with the transition from the rectangular throttling channel to circular outlet pipe and the separator wall in the throttling section, were to a large extent achieved by partial or complete blockage of cell volumes and cell faces, and hence their effective exclusion from the calculation domain. An example of this can be seen in Fig. 6.1. This basic manifold geometry was used for all four test cases. The ramp geometries, however, differed from case to case and had to be redefined for each test case. Fig. 6.1 represents the flow domain geometry for the ramp model at a ramp angle of 110° .

6.5.2 Incorporation of boundary conditions

Both the nature of the flows considered and the form of the governing equations require careful identification of boundary conditions applying to them. The boundaries to be considered are of two kinds, namely conditions appropriate to the confining walls and to the planes at inlet and outlet. The boundary information required by the finite-domain difference equations effectively takes the form of relations connecting the boundary fluxes of the entity in question to its values at nearby grid nodes (see equations 6.6 to 6.8).

Inlet boundary conditions

For the variable ramp models, the prescribed mass flow rates at $z=0$ (i.e. inlet plane) were set to 9.6, 14.4, 19.2 and 24 $\text{Kg/m}^2\text{s}$, to introduce uniform velocities in the z -direction (W_{in}) of 10, 15, 20 and 25 m/s for ramp angles of 110° , 130° , 150° and 180° respectively. These velocities correspond to mean inlet velocities measured during the hot wire studies (cf. Chapter 5). The prescribed mass flow rates are, however, some 20% lower than the measured values. This reflects the fact that the areas used to convert the prescribed sources for inclusion in the finite-difference equations are those of the nominal grid cells, before modification by porosities (i.e. blockages to modify the square cross-section to a circular inlet pipe cross-section). The reduction of 20% in mass flow rates is therefore based on the 20% reduction of cross sectional area after the modification by cell porosities.

The value of the inlet turbulent kinetic energy (K) was set to $0.006 W_{in}$, based on the velocity fluctuation measurements in the air flow tests. The turbulence energy dissipation rate (ϵ) in this region was set to

$$\epsilon = 0.09 K^{3/4} / [0.03(1/2)D_{in}] \quad (6.17)$$

where D_{in} is the diameter of the inlet pipe. This was based on the assumption that the rate of energy dissipation is controlled by the rate at which the large eddies feed energy to the small dissipative scales, which in turn adjust in size to handle this energy. Thus, if l is the macroscale of turbulence, the dissipation ϵ should scale

on K and l as follows:

$$\epsilon \propto K^{3/2} / l \quad (6.18)$$

Exit boundary conditions

At the plane of outflow (i.e. $IZ = NZ$) the relative pressure in all cells was fixed close to zero. As the fluid is assumed incompressible, all the computed pressures are relative to this pressure. No other conditions were needed in this region. This boundary condition provides the means by which realistic values of solved for variables to be brought into the domain, should an inflow be calculated in the course of the convergence process.

The wall boundary condition

The wall boundary conditions are straightforward to specify and implement when the flow is laminar. Thus, the usual no-slip conditions pertain to the velocities. The specification for turbulent flow is of the same kind, but implementation is more difficult because it is not feasible, in the present circumstances, to solve the equations in near-wall regions, where molecular and turbulence effects are of the same order of magnitude, due to the fine computational grids required. (An appreciation of the additional effort involved can be gained from Ref. 84).

This problem is circumvented by the usual practice of "bridging" the near wall regions by use of special formulae which are intended to describe the flow there and whose implications are used to modify the main equations approximately in this region.

As mentioned above, close to solid walls there are inevitably regions where the local Reynolds number of turbulence is so small that the viscous effects predominate over turbulent ones. Wall-functions have been proposed and used by many authors including Spalding (80), Wolfshtein (81), Patankar and Spalding (82) and Launder and Spalding (79), to account for these regions in numerical methods of computing turbulent flow.

The particular formulae used are of the type cited in Ref. 83 as being appropriate to one-dimensional turbulent Couette flow which obeys the "logarithmic law of the wall". They are:

$$V^+ = [\ln E y^+] \quad (6.19)$$

$$\frac{\partial K}{\partial y} = 0 \quad (6.20)$$

$$\epsilon = C_\mu^{3/4} K^{3/2} / \kappa y \quad (6.21)$$

where V^+ and y^+ are respectively the tangential velocity and normal distance in dimensionless law-of-the-wall form, and κ , E and C_μ are constants given in the Notation section.

To prescribe wall friction, the wall-function sequences in the program are triggered by values of Reynolds numbers based on the resultant velocity parallel to the wall and the distance from the wall of the grid node. Under laminar conditions ($Re < 132.25$, the value at which the laminar and turbulent wall function intersect), the wall shear stress was evaluated using a linear velocity variation between the calculated near-wall value and the prescribed wall value. For turbulent cases, the presumed velocity variation was based on the logarithmic law of the wall, and values of K and ϵ at the near-wall grid nodes are fixed at the values which would prevail there if indeed the universal logarithmic velocity profile prevailed.

In the regions where the geometry is defined by partial blockage of the cells, such as the sloping ramps and the inlet and outlet pipes, no wall boundary condition could be prescribed. This was mainly due to the exclusion of the relevant cell-faces from the calculation domain. The alternative course of action would have been to represent the geometry by complete blockage of the relevant cells, and hence introduce step-wise wall boundaries. This, however, was assumed to be less satisfactory in representing the flows under study.

6.5.3 Convergence and grid independence

In order that the solutions of the differential equation represent a satisfactory approximation, the number of grid nodes had to be

large enough for any further increase in their number to produce only insignificant changes in the values of the important predicted quantities. The only reliable way of determining how large this number should be is that of making a gradual increase of NX , NY and NZ , in successive repetitions of what is otherwise the same calculation, until the important quantities exhibit grid-independence. Increasing the number of cells, combined with the repetitive nature of the process naturally also increases the computational expense. The grid specification set out earlier was found to be close to grid independency, with little change in the values of the predicted quantities being observed at larger grid settings. These grid dependency studies were concentrated on the refinement of grids in the y - and z -directions downstream of the ramp, because the flow in this region exhibits the most pronounced variations in axial velocity and other relevant predicted quantities. The largest grid mesh used was ($NX = 7$, $NY = 25$, $NZ = 35$). The max change in the value of the relevant variables (i.e. U , V , W , K , ϵ and P) was found to be typically of the order of a few percent (i.e. less than 5%).

For ramp angles 110° and 130° , after 450 sweeps of the domain, the monitored values (downstream of the ramp) of U , V , W , K and P were found to be largely unchanging over the last 20 to 50 sweeps. Furthermore, the sum for all cells of the absolute volumetric continuity errors was found to be less than 0.1 to 0.05% of the total mass inflow (depending on the ramp angle and inlet mass flow rate). Hence, it was concluded that the solutions were reasonably well converged.

For ramp angles 150° and 180° , 350 sweeps of the calculation domain was found sufficient to produce similarly converged results.

The results of the computational simulation for the variable ramp models can be seen in Figs. 6.3 to 6.12, and will be discussed in section 6.9 in detail.

6.6 Steady flow simulation in the butterfly valve throttling system

In this section the four cases for the butterfly valve for which predictions were obtained are presented. They were modelled to simulate the actual flow conditions in the measurement studies as closely as possible at valve angles of 30° , 50° , 65° and 90° or wide-open throttle.

6.6.1 Grid arrangement and geometric configuration

The major problem concerning the prescription and simulation of the geometry was that of the complexity of the system as a whole, and its "trouser-leg" manifold geometry in particular. Unlike the variable ramp cases where a cartesian coordinate system fitted the geometry relatively accurately, neither polar nor cartesian coordinate systems appeared to provide a practical starting point from which the geometries could be simulated realistically. It was therefore decided to simplify the manifold geometry by simply neglecting the effect of the "trouser-leg" pipes representing the inlet manifold. The flow was therefore simulated through a straight pipe instead. The drawbacks are, of course, obvious but at the same time it was not expected that this would have a very significant effect on the flow further downstream of what would have been the manifold junction. This geometric simplification meant that a polar coordinate system could be used in representing the flow.

The overall geometry of the flow domain was treated as consisting of two independent parts:

- 1) The basic manifold geometry which is common for all test cases, and consists of the inlet and outlet pipe connected via a tapered section (representing the reduction in cross-sectioned area of the pipe from inlet to outlet at the manifold junction).

- 2) The butterfly valve configuration, which is different for each test case and depends on the throttle setting.

The number of grid nodes used for the butterfly valve models were 14820, 11340, 17320 and 133 for valve angles 30° (i.e. $NX = 18$, $NY = 14$, $NZ = 45$), 50° (i.e. $NX = 26$, $NY = 15$, $NZ = 38$), 65° (i.e. $NX = 26$, $NY = 18$, $NZ = 37$) and 90° or wide open throttle (i.e. $NX = 1$, $NY = 7$, $NZ = 19$), respectively. The large number of nodes used in the models was mainly due to the fine grid spacing required for the construction and simulation of the valve geometries, except in the wide open throttle case where the valve was eliminated and the flow was simulated as a two-dimensional pipe flow. Fig. 6.2 illustrates how the butterfly valve geometry and the tapered geometry of the pipe further downstream are represented by the blockage of relevant cells in the integration domain.

6.6.2 Incorporation of boundary conditions

The inlet and outlet boundary conditions were similar to those for the variable ramp models and were set at $Z = 0$ and $Z = NZ$, respectively. The prescribed mass flow rates at $Z = 0$ (i.e. inlet plane) were set to 12, 18, 24 and 30 $\text{Kg/m}^2\text{s}$, to introduce uniform velocities in the z -direction (W_{in}) of 10, 15, 20 and 25 m/s for valve angles 30° , 50° , 65° and 90° , respectively. The prescribed values of K and ξ in this region were similar to those used for the variable ramp models.

Wall boundary conditions similar to those identified in the previous section (6.5.2) were prescribed for the inlet and tapered outlet pipes. No wall boundary condition, however, was prescribed for the valve surfaces. This was mainly because the geometry of the valve was represented by complete blockage of relevant cells, which in turn resulted in step wise valve surfaces.

6.6.3 Convergence and grid independence

For a valve angle of 30° , after 400 sweeps of the calculation domain the monitored values of variables downstream of the valve were found to be unchanging over the last 25 sweeps. Furthermore, the sum for all cells of the absolute volumetric continuity errors was found to be of the order of 0.1% of the total mass inflow. Similarly converged results were achieved for valve angles of 50° and 65° with 350 sweeps. For the wide open case 100 sweeps of the calculation domain produced continuity errors of the order of 0.1% of the total mass inflow. Therefore, it was concluded that the solutions were satisfactorily converged.

No grid independence studies were carried out since the grid spacing was already highly refined as a result of the geometric requirements of the butterfly valve.

The results of the computational simulation for the butterfly valve models can be seen in Figs. 6.13 to 6.21, and will be discussed in section 6.8 in detail.

6.7 In-cylinder steady flow simulation

In this section the eight cases of in-cylinder flow for which predictions were obtained are presented, namely those for butterfly valve angles of 30° , 50° , 65° and 90° and variable ramp angles of 110° , 130° , 150° and 180° .

The main reason for separate modelling of the flows for the manifold and throttling device, the cylinder and inlet valve was that in the variable ramp cases it was not possible to accommodate the geometry of the cylinder with a cartesian coordinate system. For the butterfly valve cases it was argued that separate modelling could provide computational savings in areas such as convergence and grid independence studies.

6.7.1 Grid arrangement and geometric configuration

The number of grid nodes used for the cylinder and inlet valve models was 1782 (i.e. $NX = 9$, $NY = 9$, $NZ = 22$), for the variable ramp cases, and 5148, 9152, 10296 and 198 for butterfly valve angles of 30° (i.e. $NX = 18$, $NY = 13$, $NZ = 22$), 50° (i.e. $NX = 26$, $NY = 16$, $NZ = 22$), 65° (i.e. $NX = 26$, $NY = 18$, $NZ = 22$) and 90° or wide open throttle (i.e. $NX = 1$, $NY = 9$, $NZ = 22$), respectively, with Z, Y and X being the axial, radial and circumferential directions of the flow respectively. The large and variable number of grid nodes used in the butterfly valve cases was mainly due to the inlet boundary condition requirements, and will be explained later.

The geometry of the models consisted of a straight inlet pipe of diameter 32 mm (equal to that of the outlet pipe for the throttling models), through which flow expands around a seatless poppet valve into a 75 mm diameter cylinder (see Figs. 6.22 and 6.25). The valve lift is 8 mm and the length of the inlet pipe 57 mm, which corresponds to the distance between the outlet plane of the throttling models and the inlet plane of the cylinder (i.e. length of the inlet pipe used in the experimental studies for diverting the flow through 90° into the cylinder).

6.7.2 Incorporation of boundary conditions

The outlet boundary conditions were similar to those explained previously and were applied at $Z = NZ$, the exit plane of the cylinder.

Wall boundary conditions were prescribed for the inlet pipe wall, cylinder wall, cylinder head, valve face wall, valve stem wall and valve back wall.

The values of the relevant variables at the inlet (i.e. $Z = 0$) were obtained from the throttling models. These values were taken from the X-Y plane at $Z = 5.2 D_{in}$ downstream of the inlet plane, corresponding to the plane at which hot wire measurements were taken upstream of the cylinder. Unlike previous cases where boundary conditions were prescribed uniformly across the inlet plane, in the present cases the variation of relevant quantities had to be represented appropriately.

To prescribe these variations, chapter 5 of the GROUND (section 6.3 and Appendix A) was used to modify the source-terms. This chapter is called during the solution for each variable at the Z-slab in question ($IZ = 1$ for inlet plane), at the stage at which the source terms in the finite-domain equations are being computed. Additional sources can then be added by means of a special sub-routine. The source-terms for mass inflow and other variables are expressed in a similar way to that explained in section 6.3.5.

The values for mass inflow, velocities, K and ϵ were expressed as source-terms and prescribed for each cell at Z-slab one.

For the butterfly valve cases this was achieved by directly matching the cylinder inlet grid distribution in the x- and y-directions to that of the relevant throttling models, hence the different number of grid nodes used in the x- and y-direction. This allowed for direct transfer of variables from throttling models to in-cylinder models.

This was not possible, however, in the case of the variable ramp models, since the coordinate system used was cartesian. To overcome this, a polar coordinate mesh was fitted over the exit plane of the throttling models, and the values of relevant variables interpolated between grid nodes. The profile assumption or interpolation formulae used was that of a piece-wise-linear profile. The total mass inflow entering the domain was found to be within 2% of the inflow prescribed for the relevant variable ramp models. In the absence of any other

readily available solution this was deemed to be an acceptable compromise.

The solutions were found to converge reasonably well after a few hundred sweeps (typically 200 to 400 sweeps), with the sum for all cells of the absolute volumetric continuity errors being of the order of 0.1 to 0.05% of the total mass inflow. The results of the computational simulations for both variable ramp and butterfly valve cases can be seen in Figs. 6.22 to 6.27, and will be discussed in the following section.

6.8 Discussion of results

6.8.1 Ramp throttle cases

The distribution of velocity vectors for the ramp throttle cases can be seen in Figs. 6.3 a, b and 6.4 a, b, c. They provide a means of direct comparison with the earlier streak photographs in Chapter 4 (in particular Plates 4.1, 4.3, 4.5 and 4.7).

At high throttle settings (included angles of 110° and 130°) the gross features and their development with throttle setting are clearly reproduced and it is therefore more instructive perhaps to focus on the principal discrepancies.

For the throttle ramp at an included angle of 110° , the predicted recirculation zone behind the ramp (Fig. 6.3 a) appears to be both more elongated and displaced toward the outer wall, relative to its water analogy counterpart (Plate 4.1 A). There is also less predicted flow deflection by the sudden expansion at entry into the throttling device and correspondingly less evidence of continued deflection through the throat by the convergent wall of the ramp subsequently. This is also reflected in the near wall prediction (Fig. 6.3 b), where the recirculation zone is also slightly more elongated relative to its water analogy counterpart (Plate 4.1 B). The reduction in the size of the recirculation zone from mid-plane to near-wall plane is, however, well predicted.

For the throttle ramp at an included angle of 130° the predicted recirculation zone behind the ramp (Fig. 6.4 a) appears to be smaller and displaced more toward the outer wall (Plate 4.3 A). As in the

previous cases there appears to be less predicted flow deflection by the sudden expansion at entry. This is particularly evident on the blocked half of the throttling section.

For low throttle settings (i.e. 150° and 180°) the major discrepancies in the predicted flow field are most noticeable and significant in the case of ramp throttle at an included angle of 150° (Fig. 6.4 b). There appears to be no predicted flow separation or recirculation behind the ramp in comparison to the water analogy picture (Plate 4.5 A). For the wide open throttle case (Fig. 6.4 c) the predicted recirculation zone behind the sudden expansion at the entry appears to be slightly smaller on the induction half of the device in comparison with its water analogy counterpart (Plate 4.7 A). The predicted flow separation on the induction side of the separator wall is, however, less obvious in the water analogy pictures.

Three specific features of the present application, as distinct from the broader uncertainties regarding K- ϵ turbulence modelling, are noteworthy in this context. Firstly, the absence of a partially-developed boundary layer in the flow through the transition section into the two-dimensional ramp throttle. Secondly, the absence of wall friction on the ramp itself; and, thirdly the simplification of geometries, in areas such as the sharp edge of the ramp apex, and the separator wall profile. The overall picture which emerges seems therefore to be of stronger sensitivity to minor geometric details and wall friction, or the lack of it, on the convergent ramp (for example, in the absence of flow separation downstream of the ramp at 150°) at low throttle settings, whereas at high throttle settings the gross features are well predicted, with less sensitivity to minor geometric details and wall boundary conditions.

Quantitative comparisons between hot wire measurements and predicted velocities and turbulent intensities downstream of the restriction are shown in Figs. 6.5 to 6.10.

For a ramp angle of 110° the apparent displacement of the recirculation zone behind the ramp towards the enter wall, noted from the flow visualization study in comparison with the computational simulation, is also evident in comparison with hot wire measurements (Fig. 6.5 a to 6.7 a). The discrepancies in the predicted velocities in the lower half of the flow imply that the flow through the Perspex

model is rather more skewed than is predicted.

The velocity predictions for a ramp angle of 130° (Figs. 6.5 b to 6.7 b) appear to agree well with the measurements. There are, however, some over-predictions in the middle of the flow, and especially so at plane (4-5-6) (the mid-plane of the device).

The absence of recirculation and flow separation in the predictions for the ramp angle at 150° is also evident in the quantitative comparisons, with over-predictions of $\lesssim 20\%$ in the middle and lower half of the flow (Figs. 6.5 c to 6.7 c). The absence of wall friction on the lower wall of the throttling duct, which is mainly the result of ramp extension, can also be accounted as a contributory factor in over-prediction and discrepancies in this region.

For the wide open throttle case the predicted flow appears to be biased towards the outer wall. This is indicated by the higher predicted velocity ($\lesssim 16\%$) in the central regions of the flow (Figs. 6.5 d to 6.7 d). This could be the result of flow separation on the separator wall (Fig. 6.4 c).

The comparisons between the predicted and measured turbulence intensities are shown in Figs. 6.8 to 6.10. Major discrepancies between prediction and measurement appear at high throttle settings (110° and 130°). As illustrated in Figs. 6.8 a to 6.10 a), for a ramp angle of 110° there are over-predictions, especially in the middle and lower half of the flow (with the calculated value based on the isotropy assumption, giving $\bar{u} = \sqrt{2K/3}$). For a ramp angle of 130° (Figs. 6.8 b to 6.10 b) measured turbulence intensities are relatively well predicted in the middle of the flow. The discrepancies, however, increase near the walls and in particular in the lower half of the flow. This indicates, in addition to the absence of wall friction on the outer wall and the transition section (i.e. transition from rectangular duct to circular exit pipe), the shortcomings of K- ϵ model with its isotropic viscosity formulation, especially in the lower half of the flow, where as a result of heavy recirculation the flow remains essentially inisotropic. The predictions appear to improve for lower throttle settings, in particular in the wide open throttle case, where the mean axial velocities and turbulence intensities are relatively close to well-defined pipe flow (Figs. 6.8 c and d to 6.10 c and d).

The computer predicted distribution of mean axial velocities and their corresponding turbulence intensities for the ramp throttle models can be seen in Figs. 6.11 and 6.12, respectively. For a ramp angle of 110° the profiles of mean axial velocity (Fig. 6.11 a) confirm the relative persistence of strongly accelerated flow through the throat, produced by the two-dimensional ramp, with peak mean axial velocities occurring at the effective throat ($Z/D_{in} = 3.6$) of the order of 100 m/s ($U_z/U_o \approx 10$), whilst the peak turbulence intensities are of the order of 15 m/s ($\sqrt{2/3}K/U_o = I \approx 1.5$) (Fig. 6.12 a). The peak axial velocity is reduced by about 25% to 75 m/s at 5.8 cm downstream of the throat ($Z/D_{in} = 5.1$). This is accompanied by a reduction in peak turbulence intensity to around 9.5 m/s ($I \approx 0.6$).

There is a reduction in normalised axial velocity (U_z/U_o) at the throat from a peak of the order of 10 for a ramp angle of 110° to 5 for a ramp angle of 130° (Fig. 6.11 b). This corresponds well with the increase in the effective throat area of the device. This reduction in peak axial velocity and velocity gradients is accompanied by a reduction in peak turbulence intensity levels to values of the order of 10 m/s ($I \approx 0.7$) (Fig. 6.12 b). They appear, however, to decay less rapidly further downstream.

The absence of predicted flow separation and recirculation downstream of the ramp for the ramp angle 150° is clearly illustrated in Fig. 6.11 c, with no region of reverse flow within the induction tract. There is also a further reduction in the value of turbulence intensity, with peak values of the order of 8 m/s ($I \approx 0.4$) near the throat. For the wide open throttle case, the flow separation on the separator wall, which was clearly indicated in the velocity vector distributions (Fig. 6.4 c), and can also be seen in the distribution of axial velocities at $Z/D_{in} = 3.6$ (Fig. 6.11 d), with a region of reverse flow in the upper half of the flow, close to the wall. There is also a turbulence intensity peak in this region (Fig. 6.12 d) which may be the result of the higher velocity gradient due to the flow separation. Referring back to the comparisons with measured values, it can be assumed that these higher mean velocity gradients, in comparison to their flow measurement counterparts, were to some extent responsible for increased turbulence production and hence higher predicted turbulence intensities near the walls, especially at low throttle

settings (i.e. ramp angles of 150° and 180°).

6.8.2 Butterfly valve cases

The distribution of velocity vectors for the butterfly valve cases can be seen in Figs. 6.13 a to d. They provide a means of direct comparison with earlier streak photographs in Chapter 4 (Plates 4.9, 4.11, 4.13 and 4.15).

The appearance of the flow past the butterfly valve is quite plausible simulated for the butterfly valve setting at 30° (Fig. 6.13 a). The location of the dividing streamline on the upstream face of the valve and the extent of the recirculation downstream appear to agree relatively well with its water analogy counterpart (Plate 4.9 B'). There is a significant reduction in the extent of recirculation downstream of the valve for the case of valve angle 50° (Fig. 6.13 b). Its shape and extent is however smaller in comparison with the water analogy case (Plate 4.11 B'). The velocity vector distributions downstream of the valve at 65° (Fig. 6.13 c) show no indication of flow separation or recirculation on the downstream face of the valve. This appears to agree well with the flow visualization observations (Plate 4.13 B').

The recirculation downstream of the valve is modest, even at high throttle settings, in comparison to their ramp throttle counterparts, and relatively well-defined pipe flow is restored within a few centimeters downstream of the valve (for example, 7 cm or 1.8 diameter for the valve angle of 30°). This is also reflected in the quantitative comparisons between hot wire measurements and computer predictions of velocities and turbulence intensities downstream of the valve (i.e. 3.7 diameter) (Fig. 6.14 to 6.19).

Overall the predicted velocity and turbulence intensity ($\sqrt{2K/3} / U_0$) profiles agree well with the hot wire measurements. That is, taking into consideration that the geometry of the manifold was simplified to a straight pipe for computer modelling purposes. What discrepancies there are appear to be mainly the direct result of the manifold junction, or its absence in case of the predictions, upstream of the plane. With the exception of the wide open throttle case, there appears to be slight over prediction of mean axial velocity

through plane (1-4-7) and plane (2-5-8) (i.e. where plane (1-4-7) corresponds to the manifold junction side of the flow and plane (2-5-8) to the mid-plane of the flow), whereas velocities through the plane (3-6-9) are consistently under-predicted (Figs. 6.14 to 6.16). This could be the result of flow separation and recirculation on the junction side of the induction tract, which was clearly indicated in the flow visualization studies. Whereas in the Perspex model the flow would be biased to one side of the induction tract as a result of flow separation at the manifold junction, in the computer models the flow is assumed symmetrical about the mid-plane of the valve. Even for the wide open throttle case over-predictions are more prominent in planes (1-4-7) and (2-5-8) in comparison to plane (3-6-9).

Turbulence intensities ($\sqrt{2K/3}$) are relatively well predicted for valve angles of 50° and 65° (Fig. 6.17 to 6.19), with some over-prediction in the lower half of the flow. These discrepancies, however, appear to become more significant for the two extreme cases, the valve angle 30° and wide open throttle cases.

Taking into consideration the geometric and boundary condition sacrifices which had to be made in simulating the flows, notably, step wise valve geometry, no wall friction on valve surfaces and the straight pipe representation of the manifold, the predictions appear to provide a reasonable simulation of the induction flow both qualitative and quantitative.

The distribution of mean axial velocity and turbulence intensities along the induction tract can be seen in Figs. 6.20 and 6.21 respectively. For a valve angle of 30° the peak mean axial velocity is of the order of 100 m/s ($U/U_\infty \approx 10$), occurring in the effective throat of the device ($Z/D = 15$) (Fig. 6.20 a). Whilst the magnitude of the peak velocities in and around the effective throat area are similar to those recorded for the ramp throttle counterpart (for example, ramp angle 110°), the peak turbulence intensities of 20 m/s ($I \approx 2.0$) are some 25% higher in this region (Fig. 6.21 a) than the ramp throttle case. The recirculation downstream of the butterfly valve appears, however, to be modest in extent and relatively well-defined pipe flow is restored within a few centimeters of the valve, and fully within 13 cm (3.5 diameter). This is accompanied by a rapid decay in turbulence intensity levels, from values of the order

of 20 m/s at the valve to 4 m/s ($I \approx 0.4$) near the exit (i.e. $Z/D = 5.0$), which is significantly less than the 9.5 m/s predicted for its ramp throttle counterpart.

For valve angles of 50° and 65° the rate at which relatively well-defined pipe flow is restored is very similar to those seen for a valve angle of 30° . The turbulence intensities, however, appear to decay less rapidly along the induction tract to values of the order of 5 m/s ($I \approx 0.35$) and 6 m/s ($I \approx 0.3$) near the exit plane (i.e. $Z/D = 5.0$) for valve angles of 50° (Fig. 6.21 b) and 65° (Fig. 6.21 c), respectively.

A well-defined pipe flow is naturally retained throughout the induction tract for Figs. 6.20 d and 6.21 d given the absence of the butterfly valve in the wide open flow domain.

6.8.3 In-cylinder cases

The mid-plane distributions of velocity vectors on air admission into the cylinder for ramp throttle cases can be seen in Fig. 6.22 a to d.

The flow distortion, as a result of the recirculation behind the ramp, persists within the induction tract as far as the inlet valve for ramp angles of 110° (Fig. 6.22 a) and 130° (Fig. 6.22 b), where it is manifest as swirl (Figs. 6.23 a and b). This is accompanied by high turbulence intensities of up to around 15 m/s (i.e. $I \approx 1.5$) and 12 m/s (i.e. $I \approx 0.8$) at the plane of the inlet valve, near the cylinder head, as shown in Figs. 6.24 a and b, respectively.

For lower throttle settings (i.e. 150° and wide open throttle) the flow distortion within the induction tract and near the valve is less prominent (Figs. 6.22 c and d), with little or no swirl in the traverse plane of the inlet valve (Figs. 6.23 c and d). There is also a reduction in the turbulence intensity levels to around $I \approx 0.4$ and 0.3 for ramp angles of 150° (Fig. 6.24 c) and 180° (Fig. 6.24 d), respectively.

Whilst at high throttle settings the two-dimensional ramp produces a pronounced asymmetric flow pattern, reflected in both the corner vortex and within the body of the chamber, the butterfly valve appears to lead to an essentially symmetric flow within the induction tract (Fig. 6.25 a to d), irrespective of the throttle setting.

This is also reflected in the distributions of radial velocities approaching the inlet valve (Fig. 6.26 a to d), whereas, in contrast to the alternative throttling device, there appears to be no indication of swirl at any throttle setting. There is also a marked reduction in turbulence intensities, especially at high throttle settings, from values of the order of 15 m/s ($I \approx 1.5$) and 12 m/s ($I \approx 0.8$) for ramp angles 110° and 130° , to 6 m/s ($I \approx 0.6$) and 8 m/s ($I \approx 0.5$) for valve angles of 30° (Fig. 6.27 a) and 50° (Fig. 6.27 b), respectively. The turbulence intensity levels for lower throttle settings, however, appear to be of the same order for both systems.

These last in-cylinder effects were not evidenced by the water analogy experiments in such convincing detail, although they were indicated by some flow bias around the inlet valve at high ramp throttle settings.

6.9 Concluding remarks

The stationary turbulent flow through the alternative induction system throttling devices -a butterfly valve and a variable geometry ramp restriction- have been modelled computationally and compared with water analogy flow visualization and hot wire flow measurements. The essential flow field characteristics are satisfactorily reproduced, including in particular the extent of the recirculation zone in the lee of the throttle and the relative persistence of turbulence generated downstream for varying throat apertures. The limited quantitative comparisons with hot wire velocity measurements lend further support to the more detailed aspects of the computational predictions.

The computer predictions, however, did suffer from a number of shortcomings, notably:

- * Isotropic viscosity formula assumption in the K- ϵ model or turbulence.

- * Absence of partially-developed boundary layer and wall friction in partially blocked regions of flow.

- * General geometric discrepancies and simplifications in the computational simulation of the flows.

Overall, the predictions appear to provide a reasonable diagnostic aid for the evaluation of the two alternative throttling systems.

In the context of engine performance the following broad conclusions could be made:

- The extent and strength of the recirculation and mixing zone downstream of the throttle restriction is much more pronounced for the ramp throttle device, in comparison with the more conventional butterfly valve system. This is particularly apparent at high throttle settings.

- The stronger recirculations associated with the variable ramp system result also in high levels of turbulence within the induction tract, which can lead to better mixture preparation and more homogeneous air-fuel charge.

- Whilst the butterfly valve leads to an entirely symmetrical flow, the two-dimensional ramp throttle produces a pronounced asymmetric pattern, essentially as a result of persistent highly accelerating flow within the induction tract as far as the inlet valve and beyond, where it is manifest as swirl.

- At high throttle settings (usually associated with low engine speeds and low charge velocities past the inlet valve) the variable ramp throttling system can improve combustion and the extent of the lean mixture limit of stable combustion, in comparison with the butterfly valve system, by improving mixture formation as well as increasing gas motion, in terms of both swirl and higher gas velocities past the inlet valve inside the cylinder.

In this chapter it has been clearly demonstrated that numerical simulation offers very considerable potential as a diagnostic aid in internal flow design. This is even more true of the non-stationary flows arising in these components in engine operation, where the very absence of experimental observation makes the numerical simulation uniquely valuable, although also severely restricting the opportunities for validation. The following chapter will, therefore, predominantly deal with the initial steps required in developing a realistic unsteady computer simulation of such engine flows.

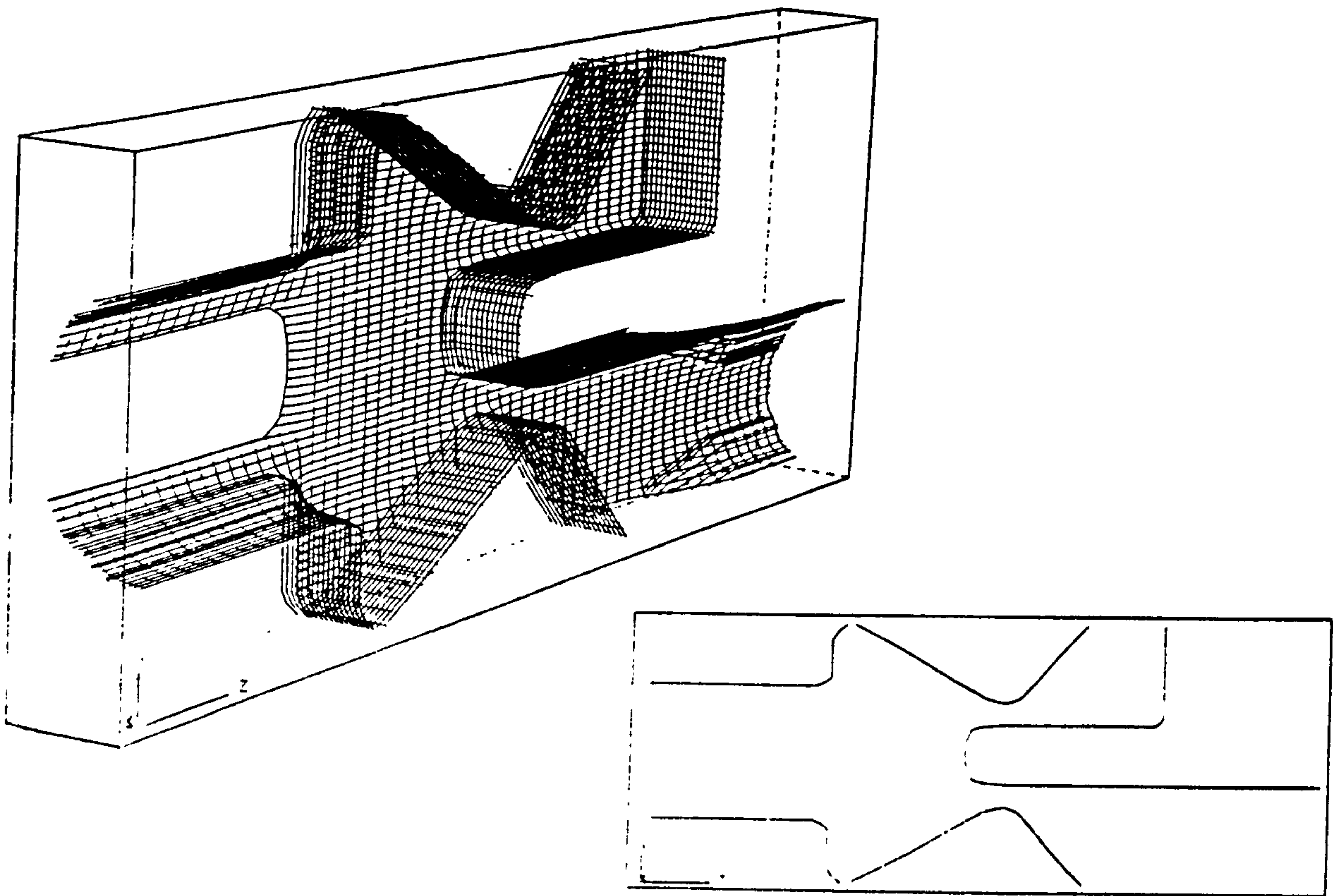


Fig. 6.1 Simulation of variable ramp geometry in rectangular cartesian coordinates.

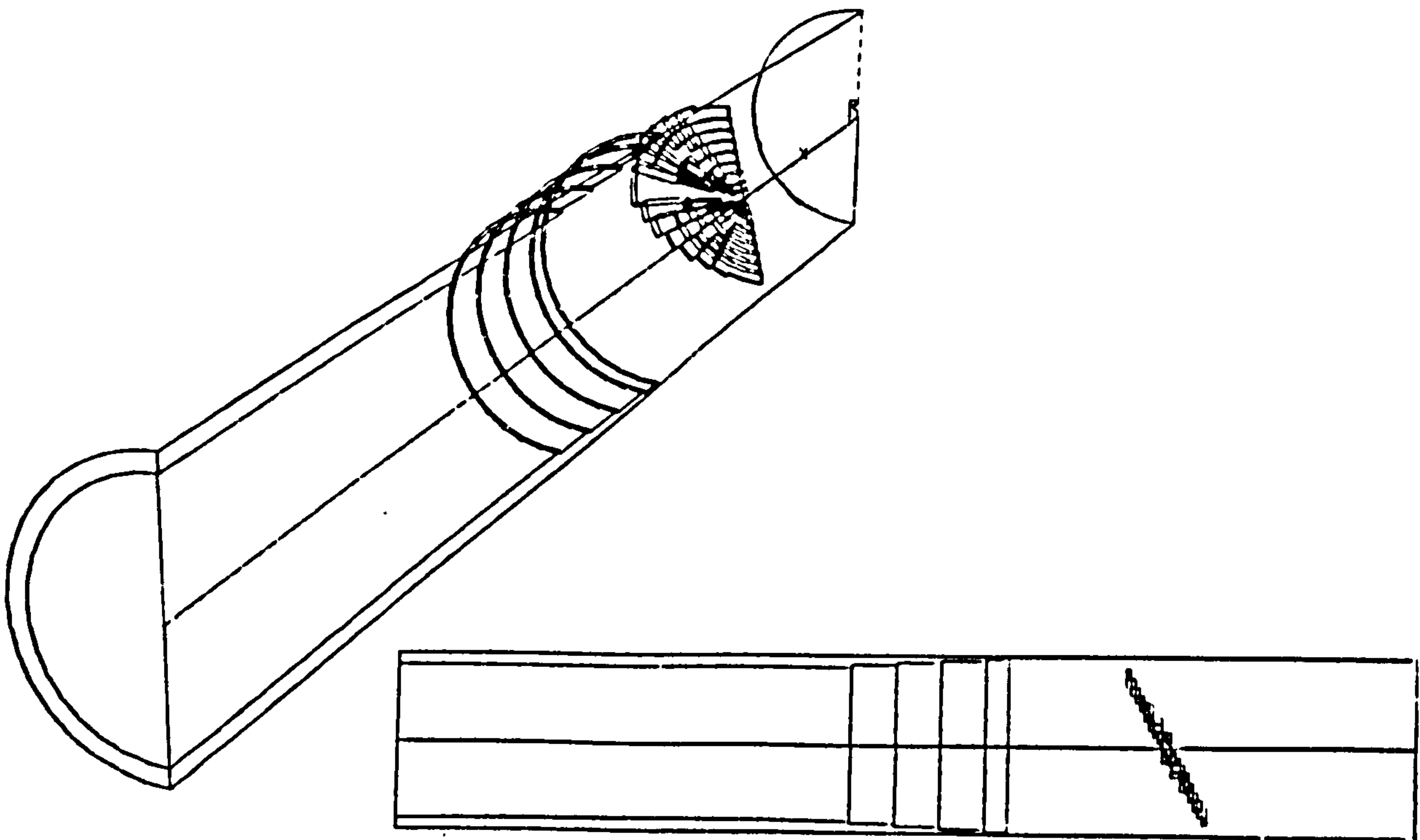
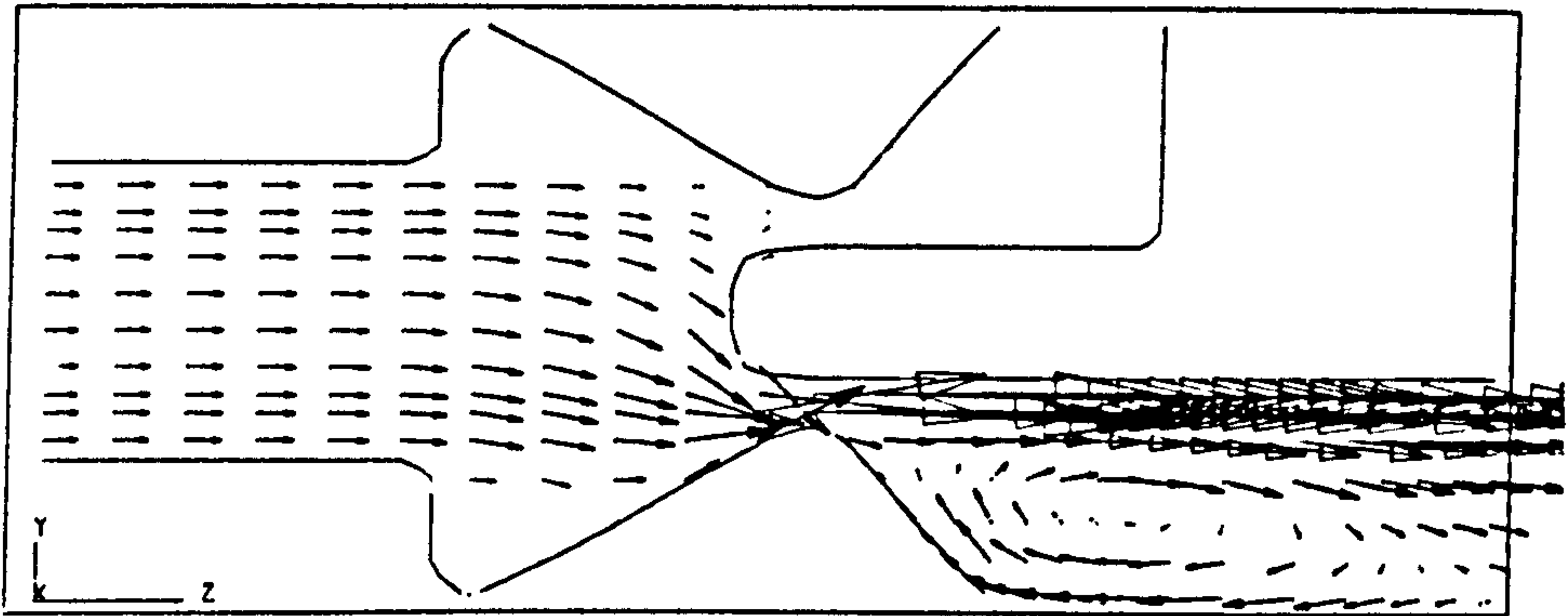
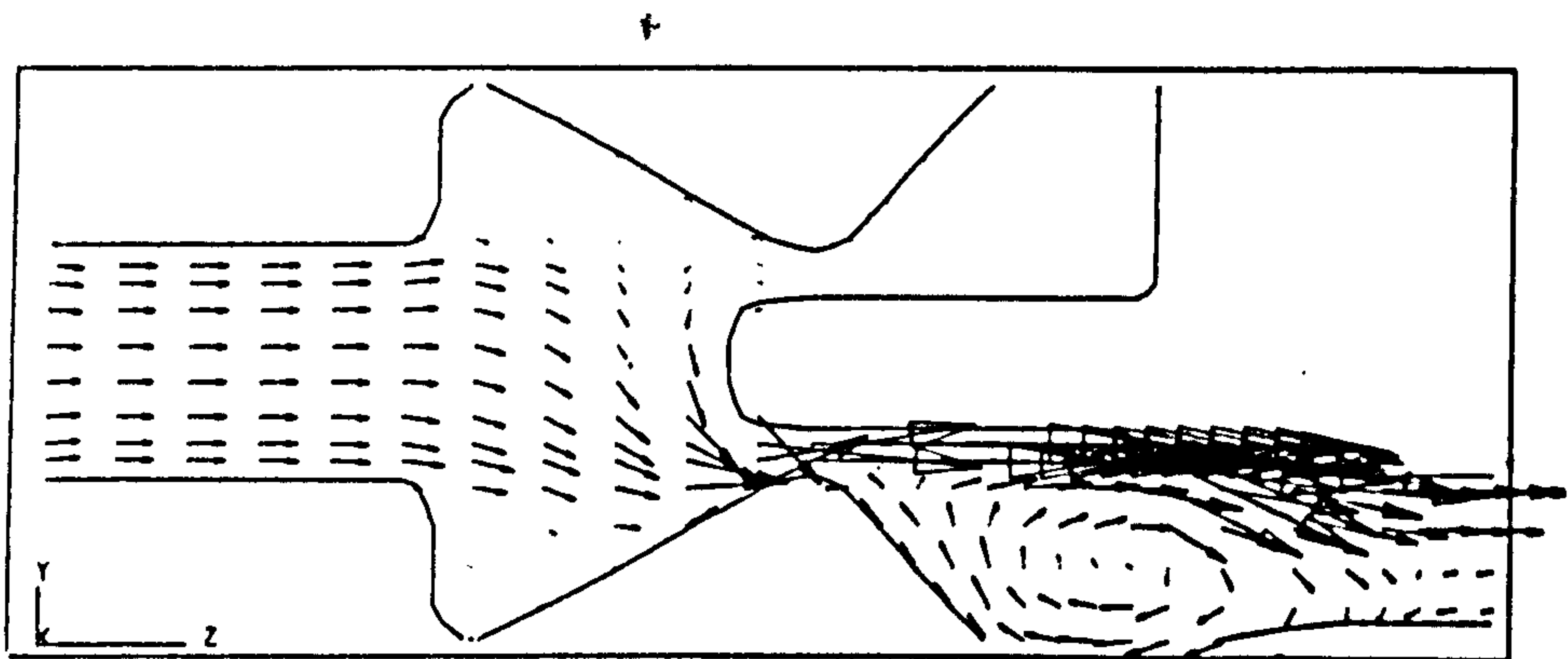


Fig. 6.2 Simulation of butterfly valve geometry in cylindrical polar coordinates.

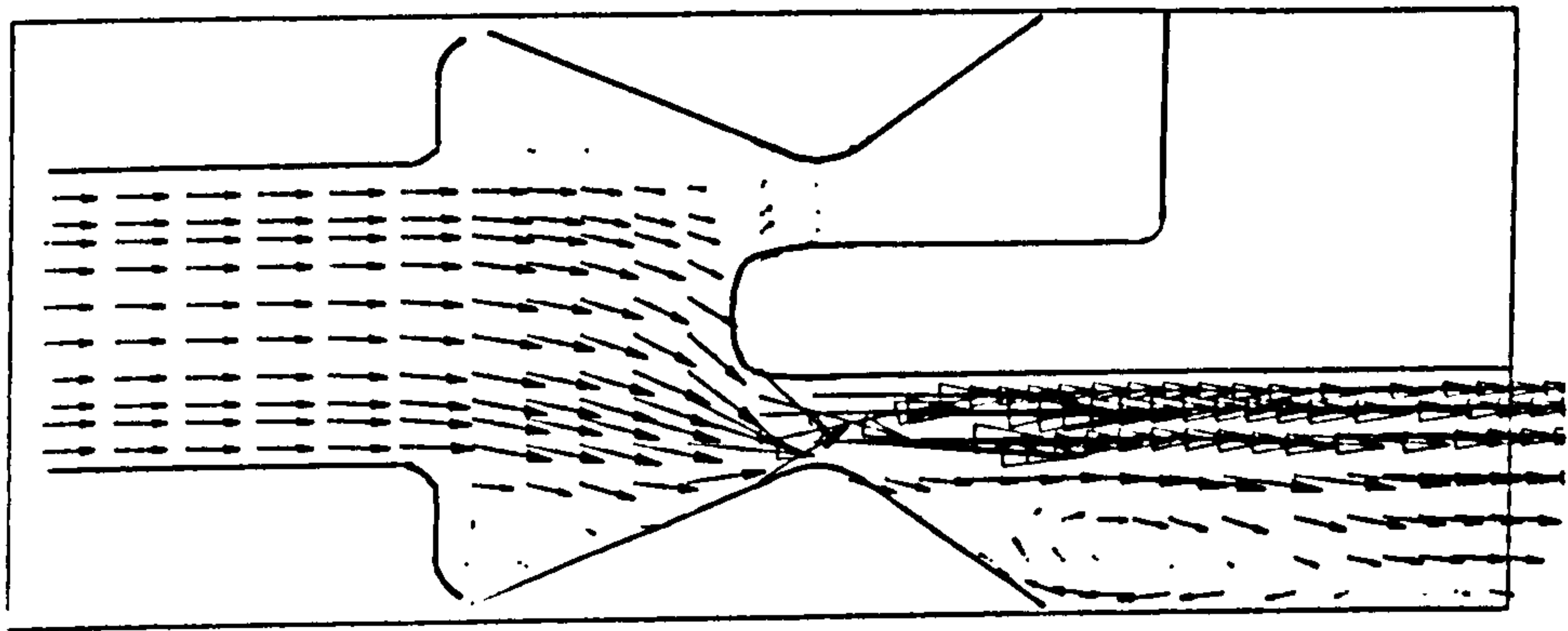


(a)

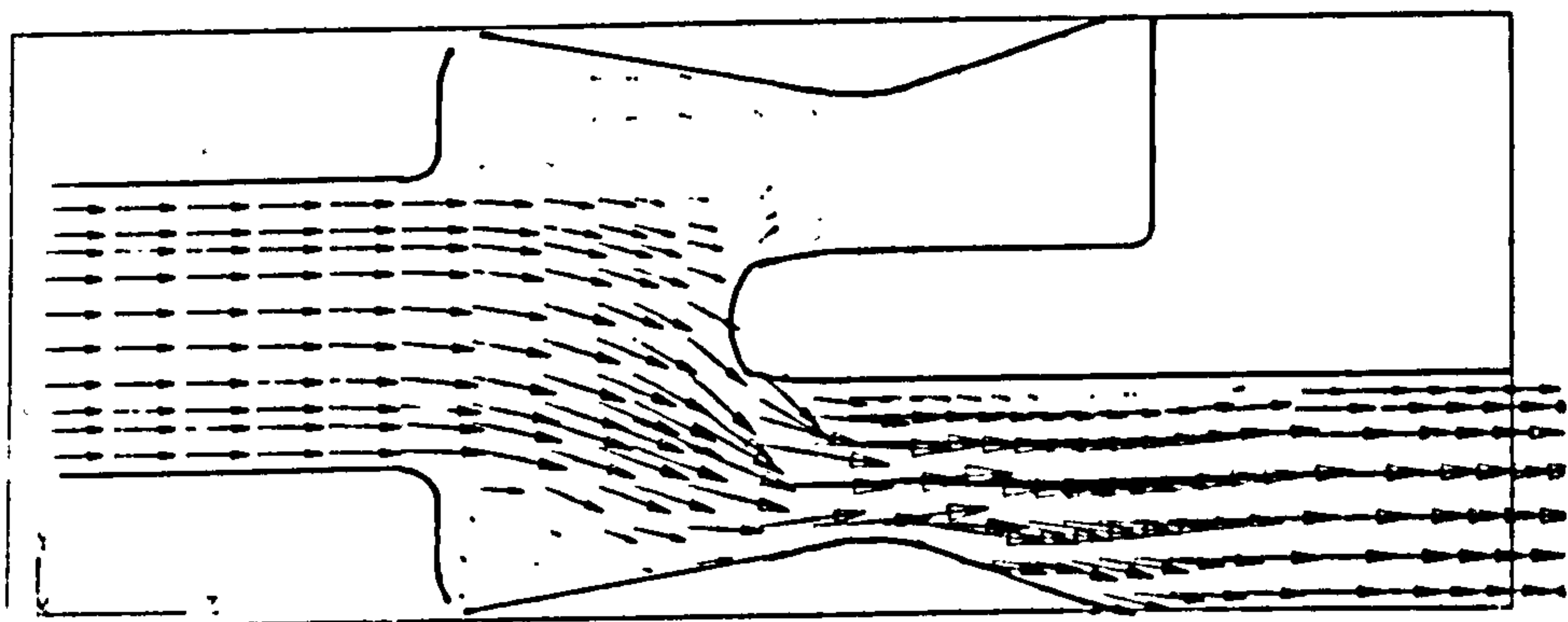


(b)

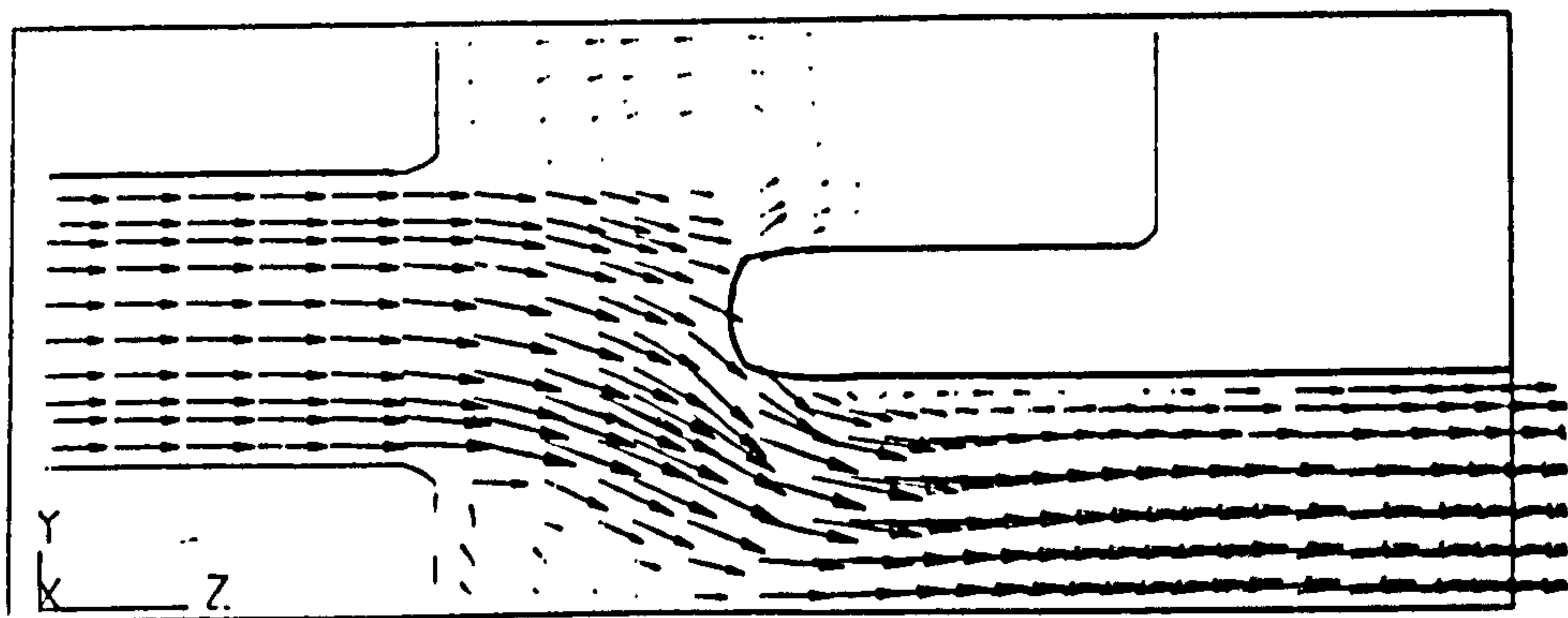
Fig. 6.3 Distribution of velocity vectors in the ramp throttle from the computer simulation for included angle of 110° at mid-plane (a) and near wall plane (b).



(a)



(b)



(c)

Fig. 6.4 Distribution of velocity vectors in the ramp throttle mid-plane from the computer simulation at included angles of 130° (a), 150° (b) and 180° (c), respectively.

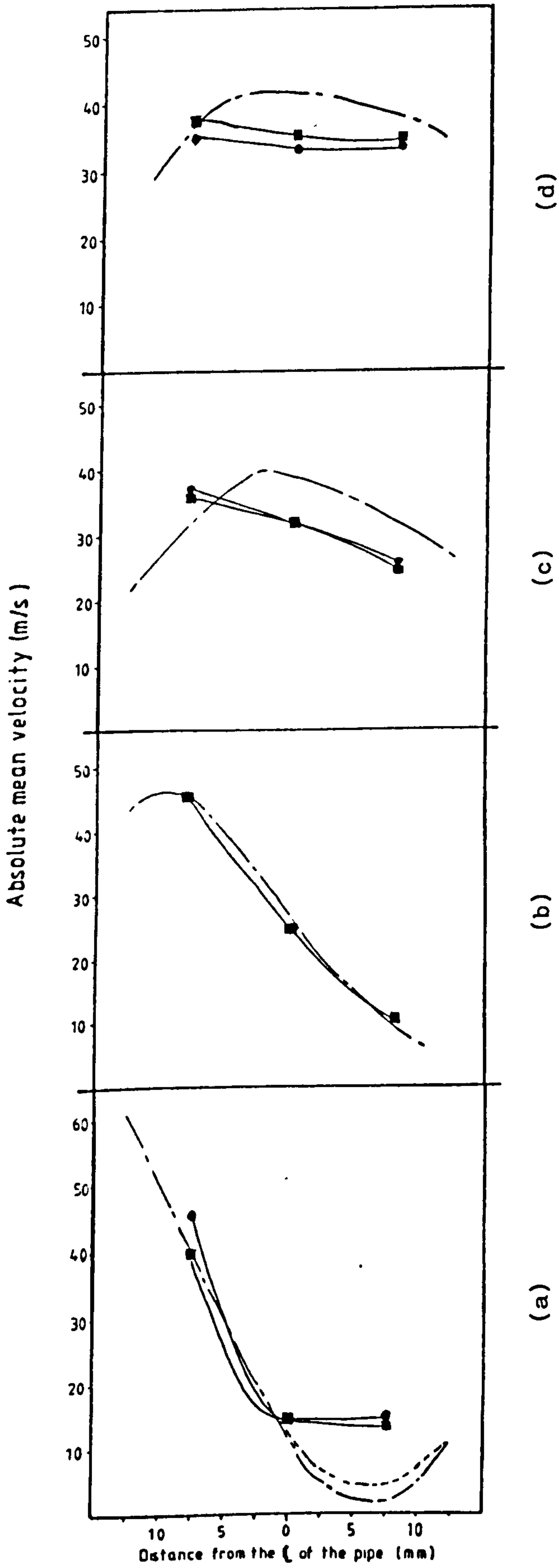
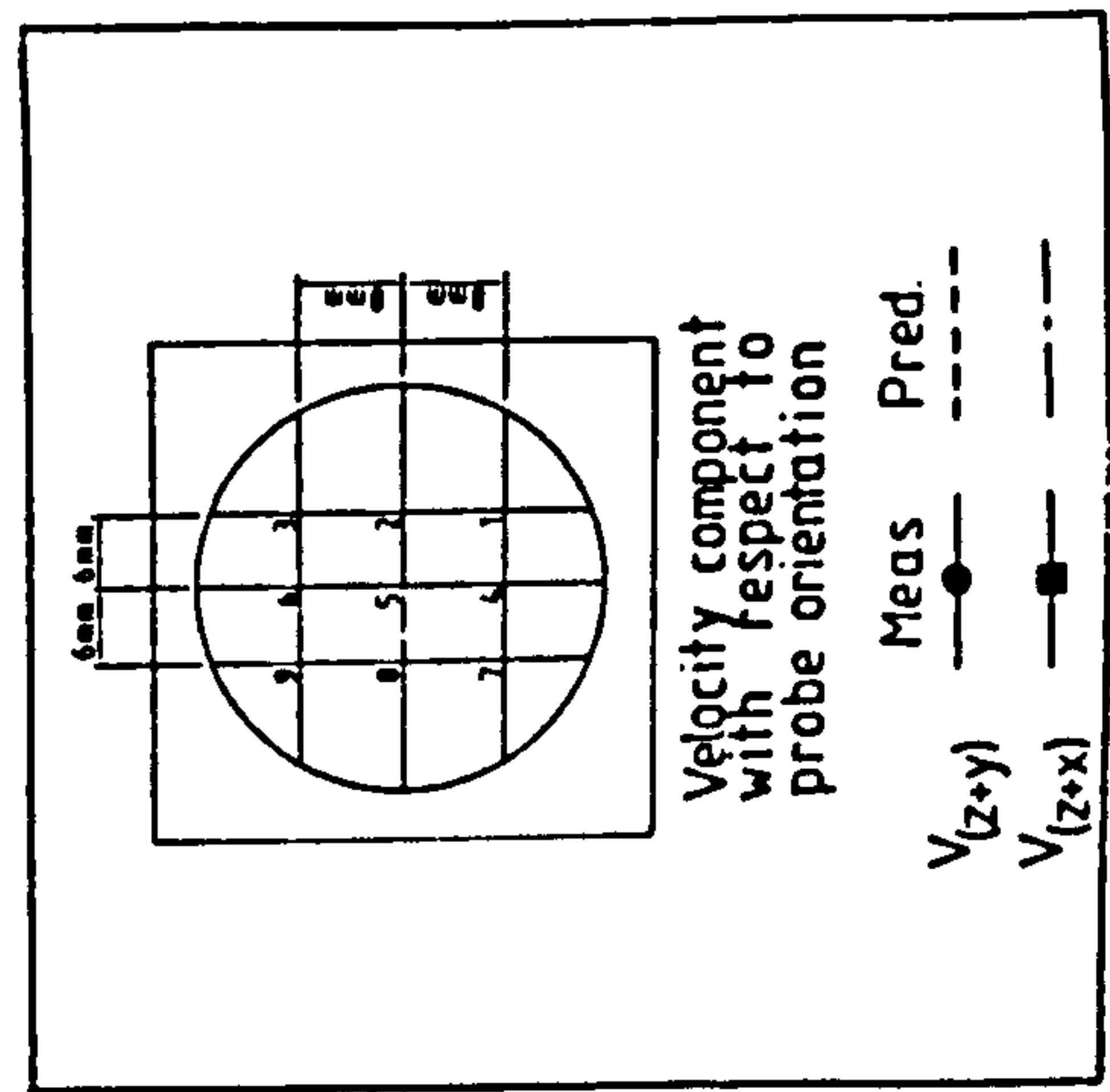


Fig. 6.5 Distribution of absolute mean velocity at plane (1-2-3) for ramp throttle at 110° (a), 130° (b), 150° (c) and 180° (d).



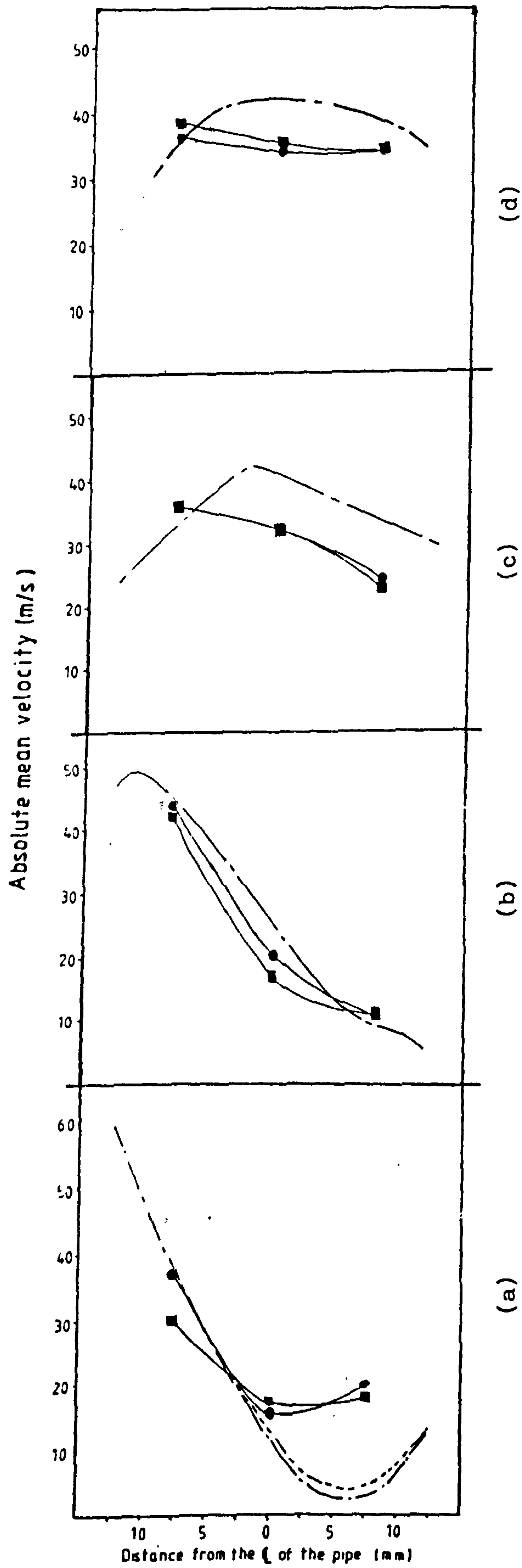
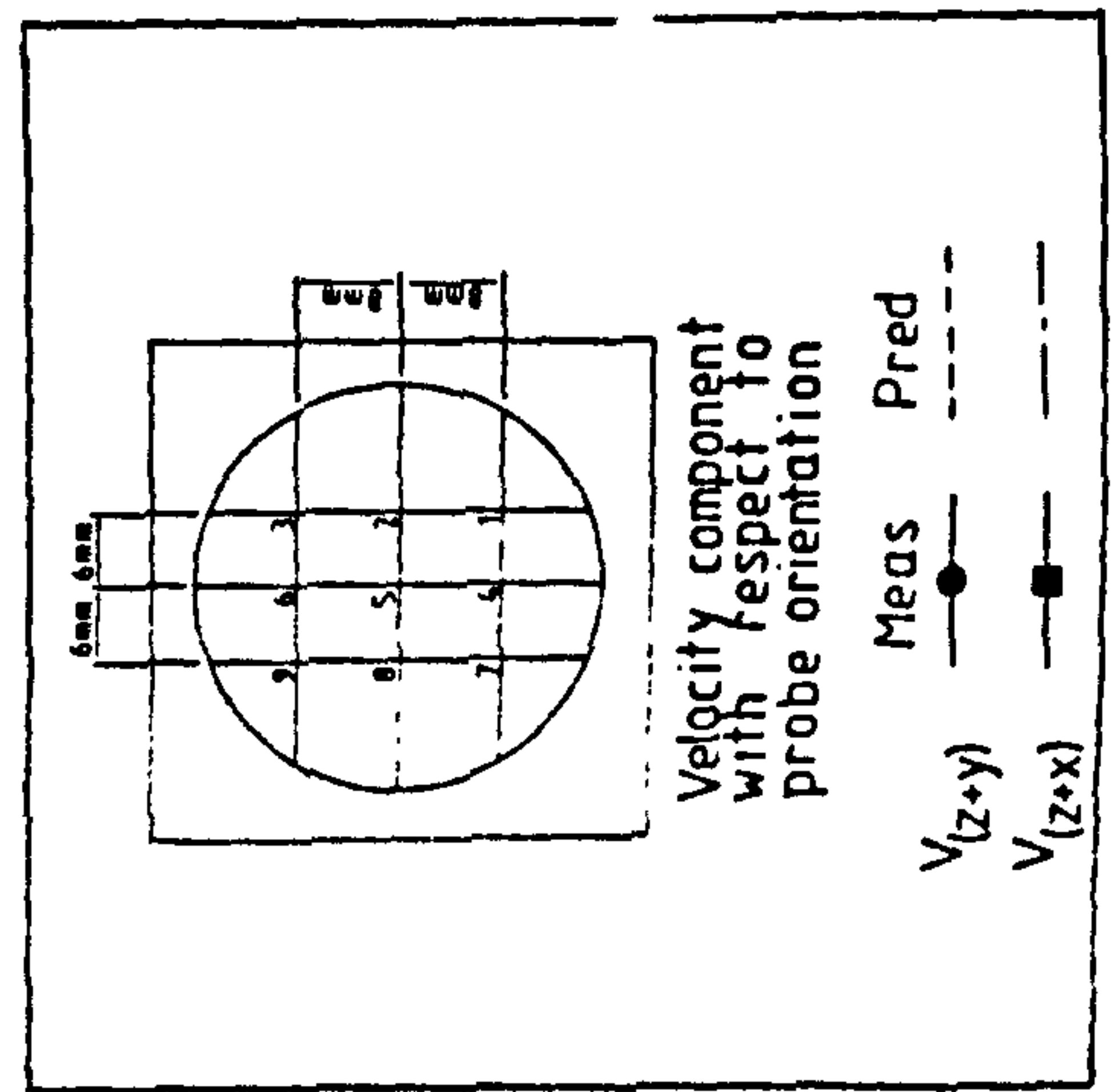


Fig. 6.6 Distribution of absolute mean velocity at plane (4-5-6) for ramp throttle at 110° (a), 130° (b), 150° (c) and 180° (d).

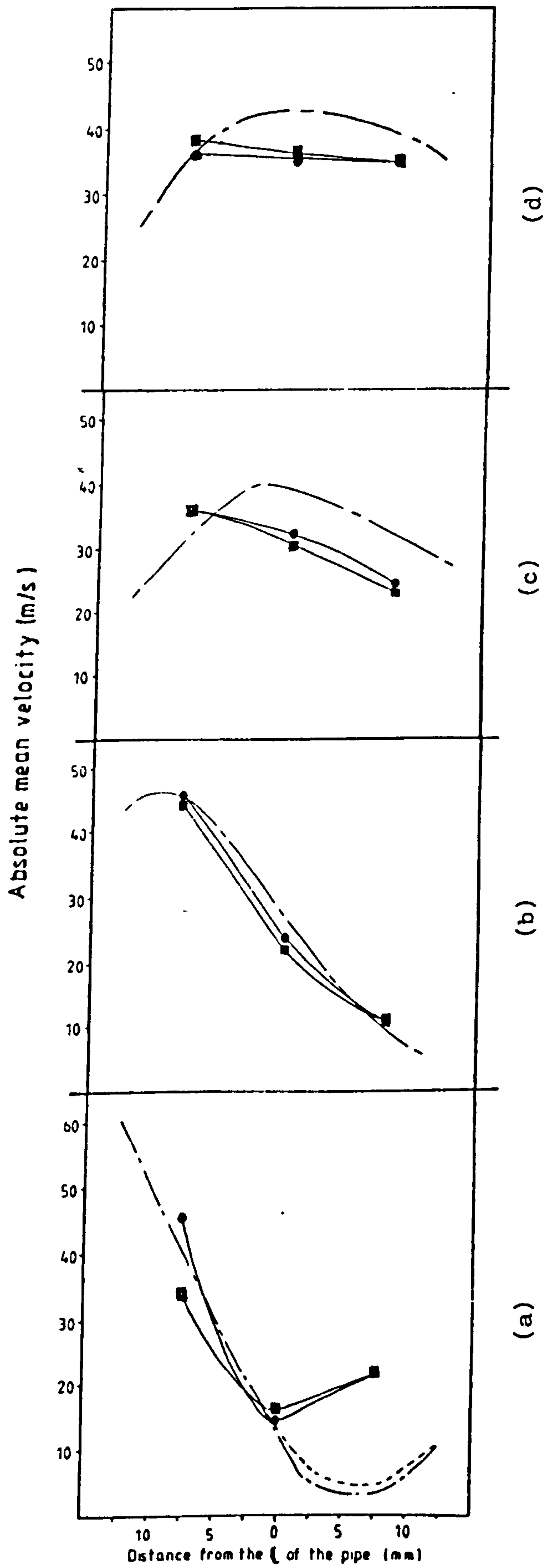
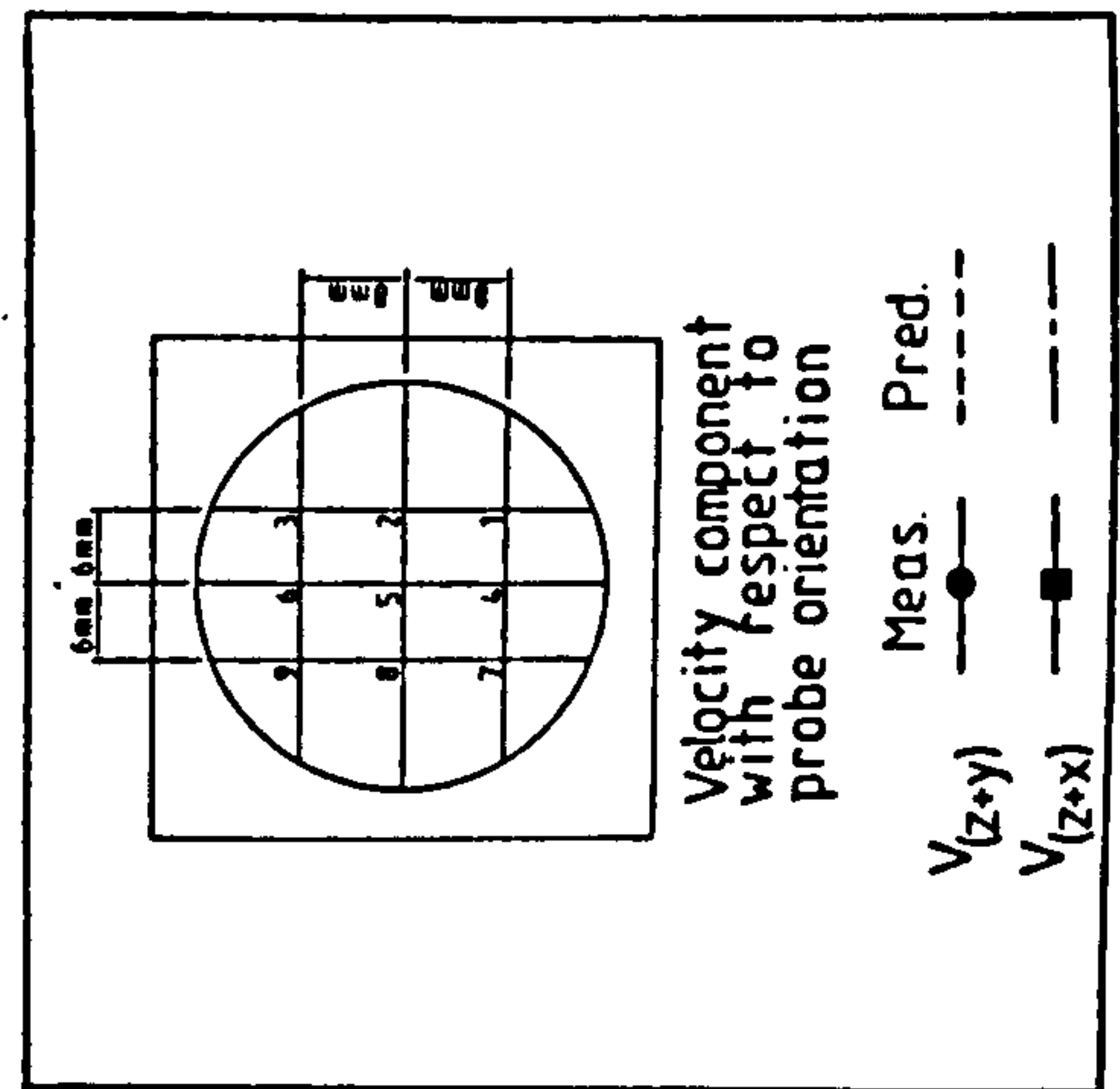


Fig. 6.7 Distribution of absolute mean velocity at plane (7-8-9) for ramp throttling at 110° (a), 130° (b), 150° (c) and 180° (d).

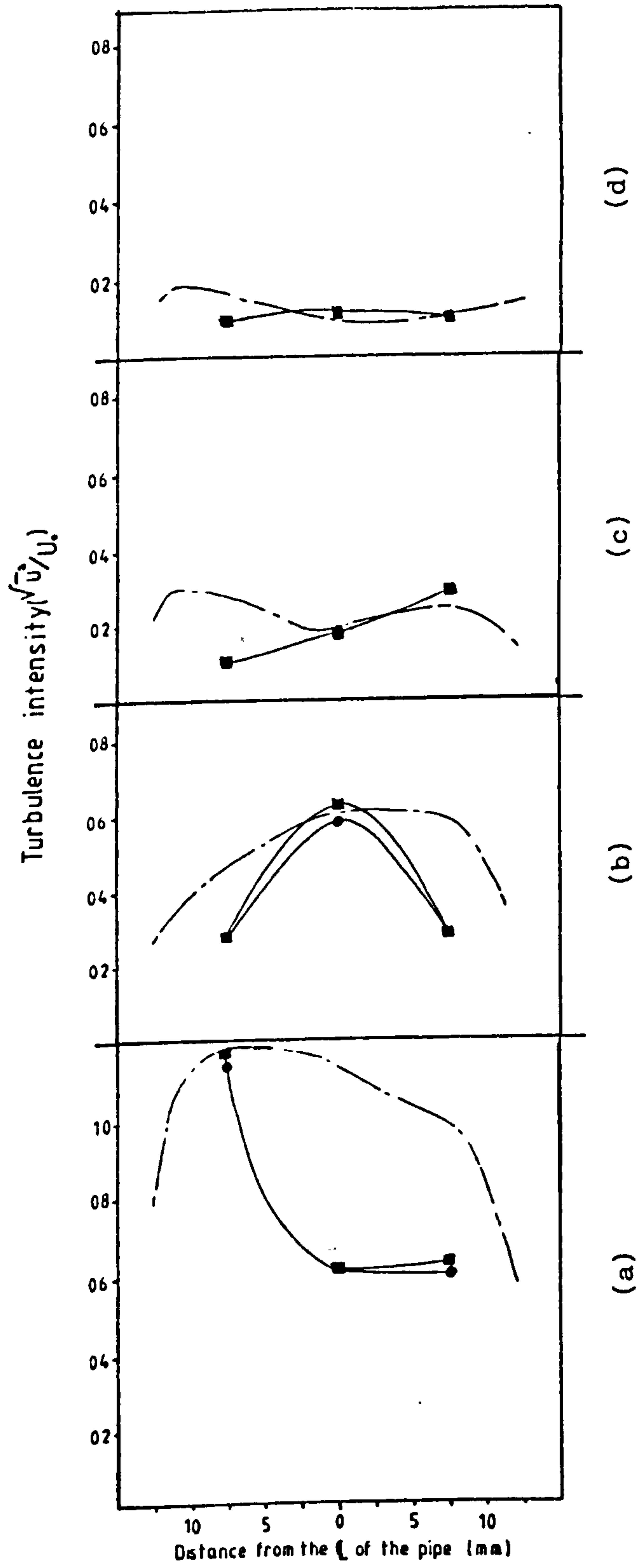
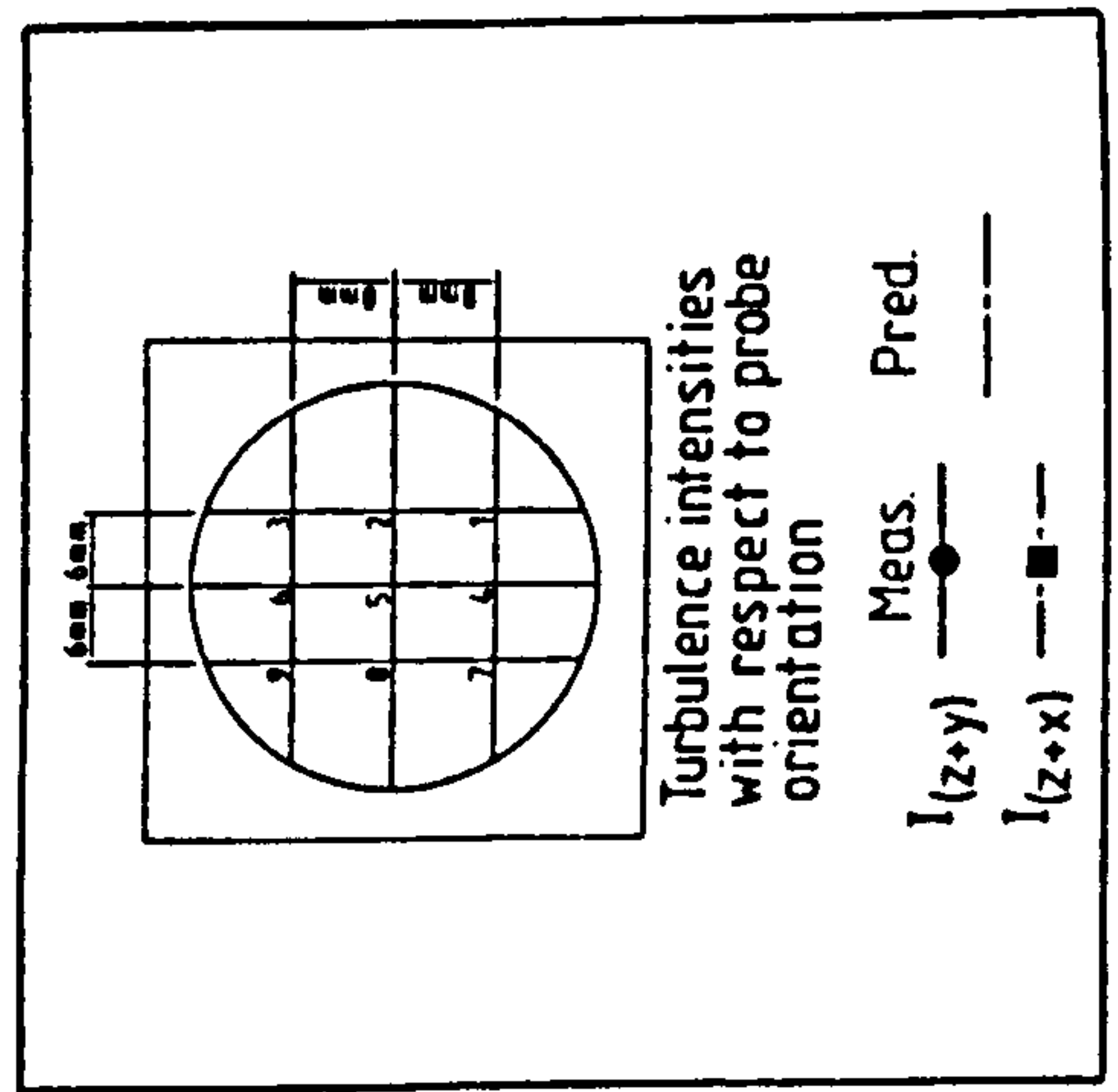


Fig. 6.8 Distribution of turbulence intensity ($u^2 = \frac{2}{3} K$), at plane (1-2-3) for ramp throttle at 110° (a), 130° (b), 150° (c) and 180° (d).



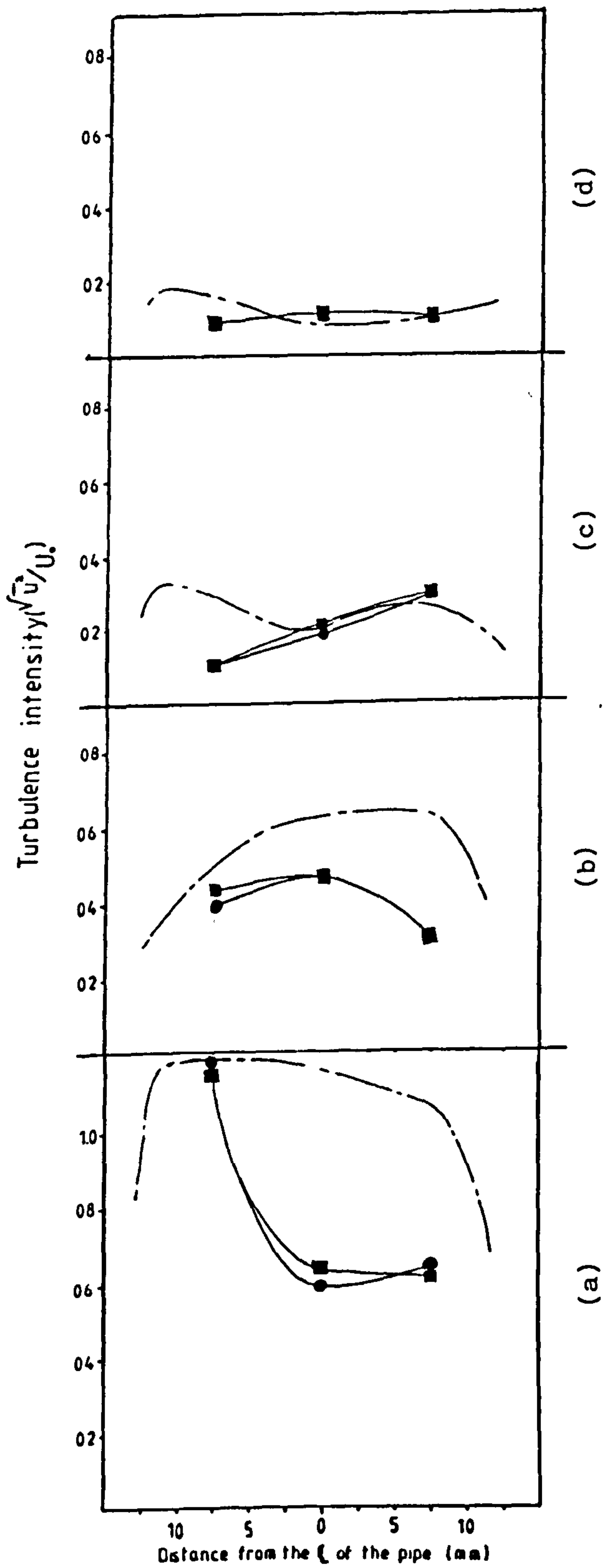
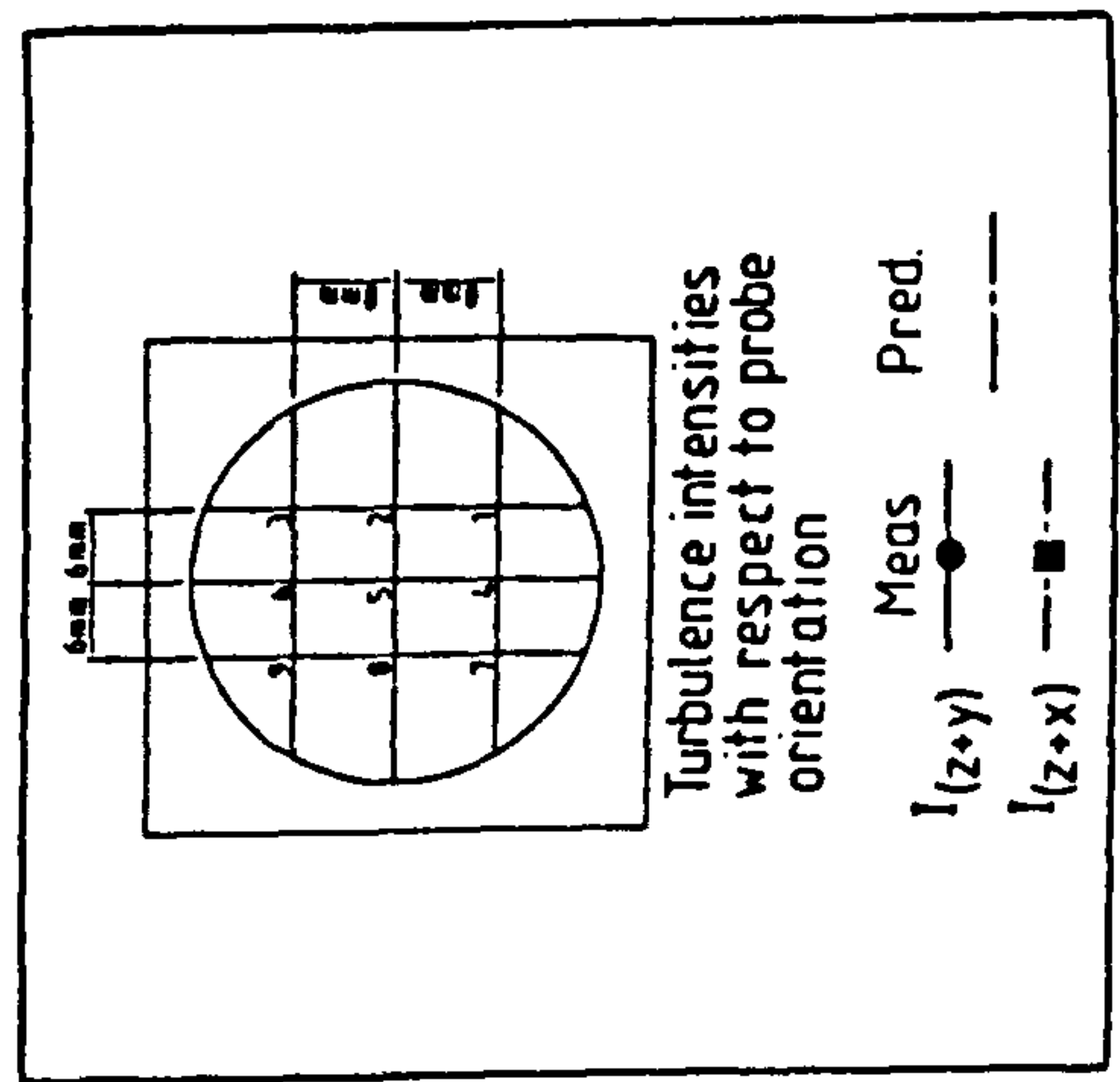


Fig. 6.9 Distribution of turbulence intensity ($u^2 = \frac{2}{3} K$), at plane (4-5-6) for ramp throttle at 110° (a), 130° (b), 150° (c) and 180° (d).



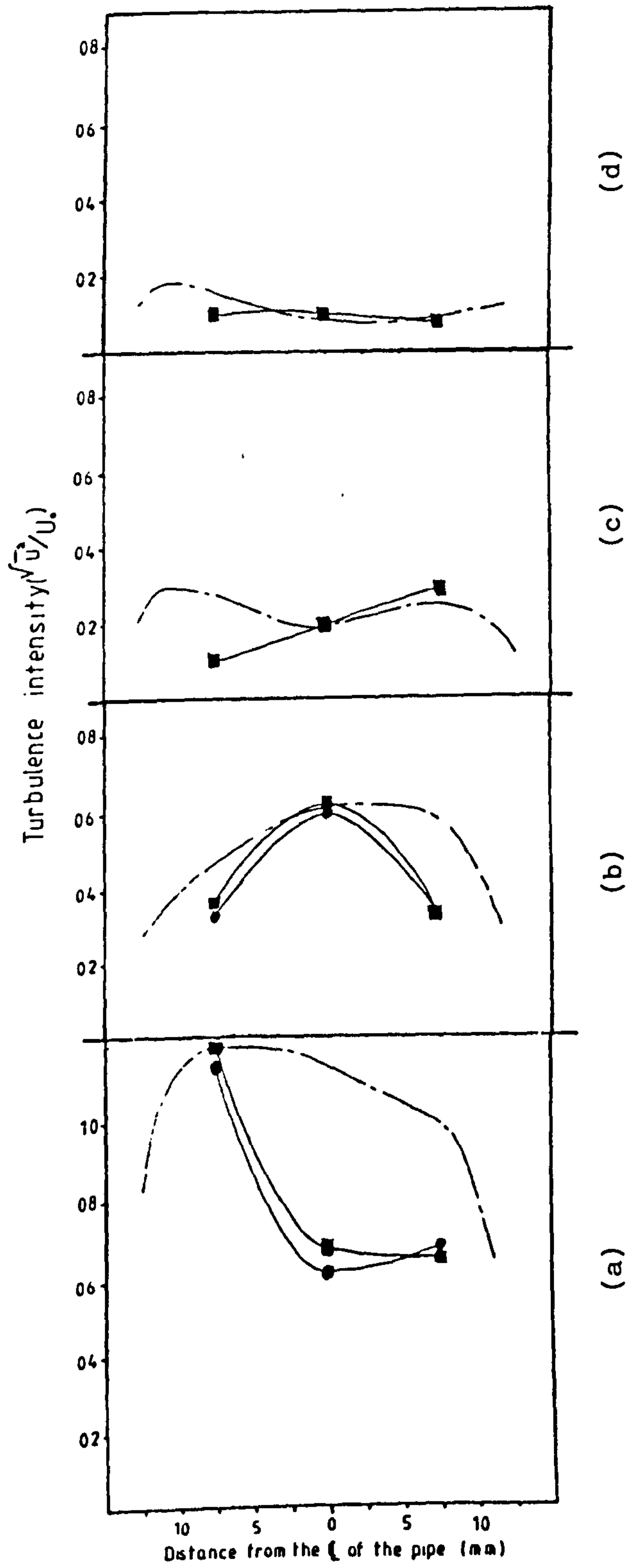
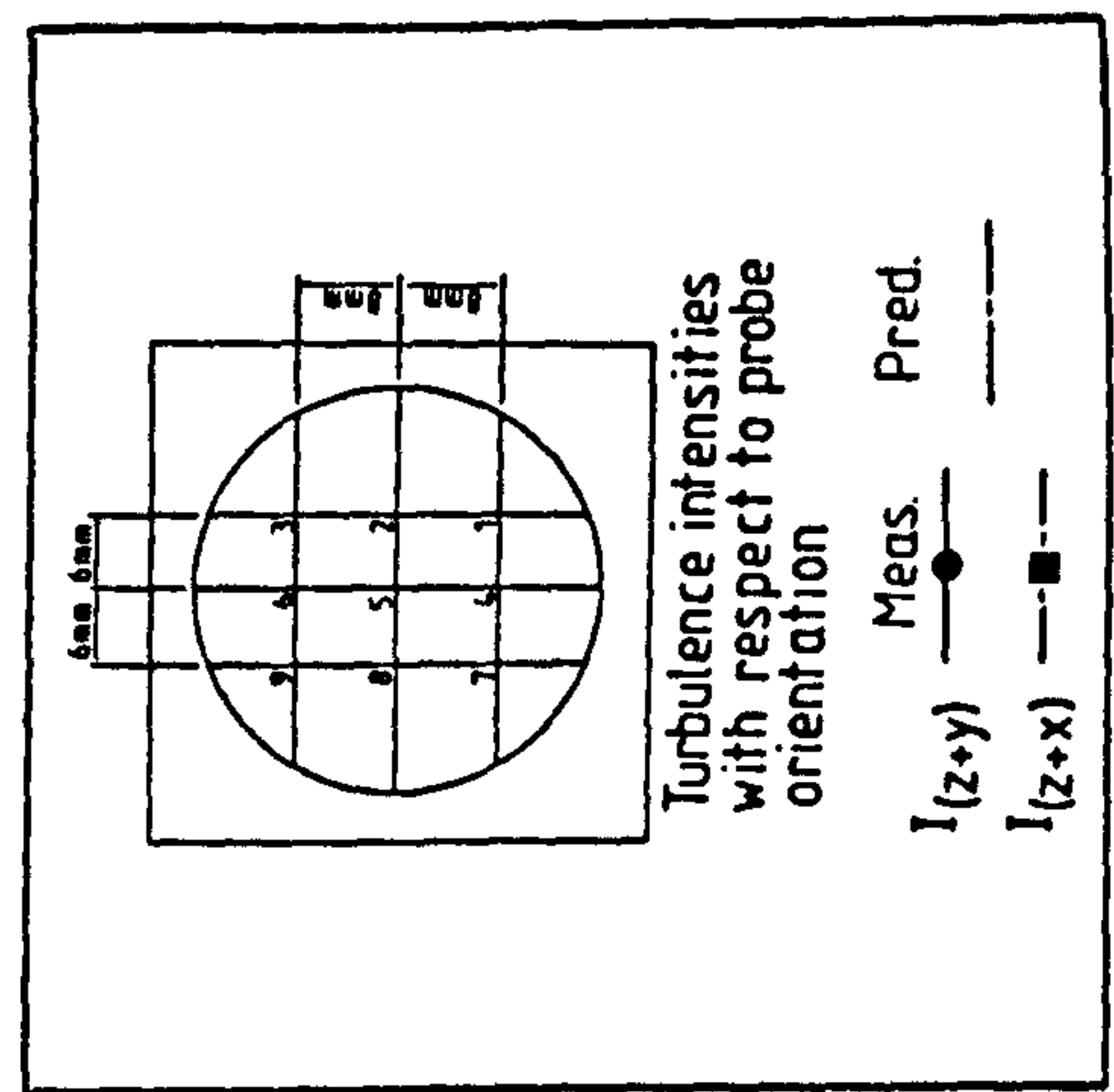


Fig. 6.10 Distribution of turbulence intensity ($u^2 = \frac{2}{3} K$), at plane (7-8-9) for ramp throttle at 110° (a), 130° (b), 150° (c) and 180° (d).

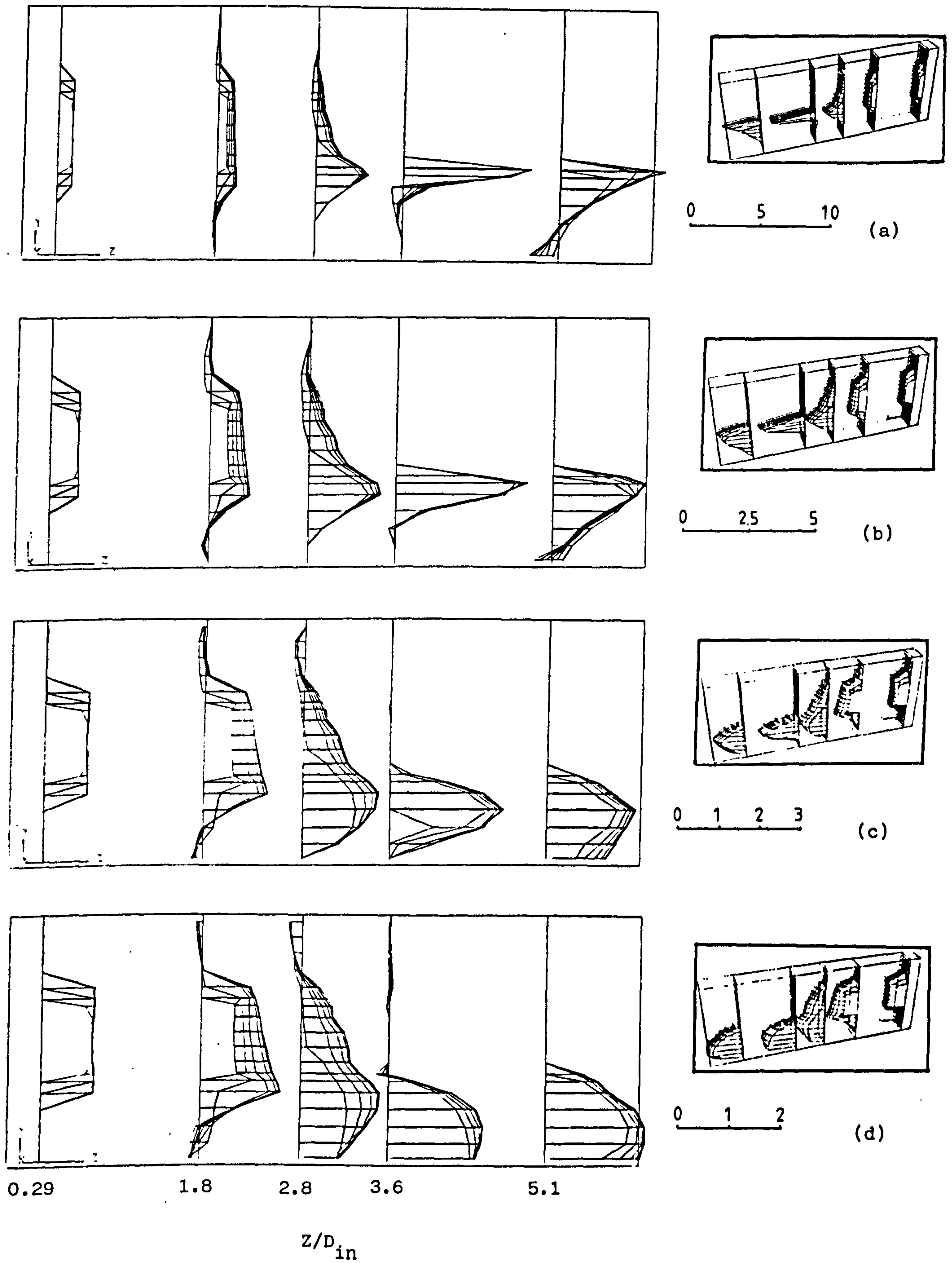


Fig. 6.11 Distribution of mean axial velocity (U_z/U_0) for the ramp throttle at 110° (a), 130° (b), 150° (c) and 180° (d).

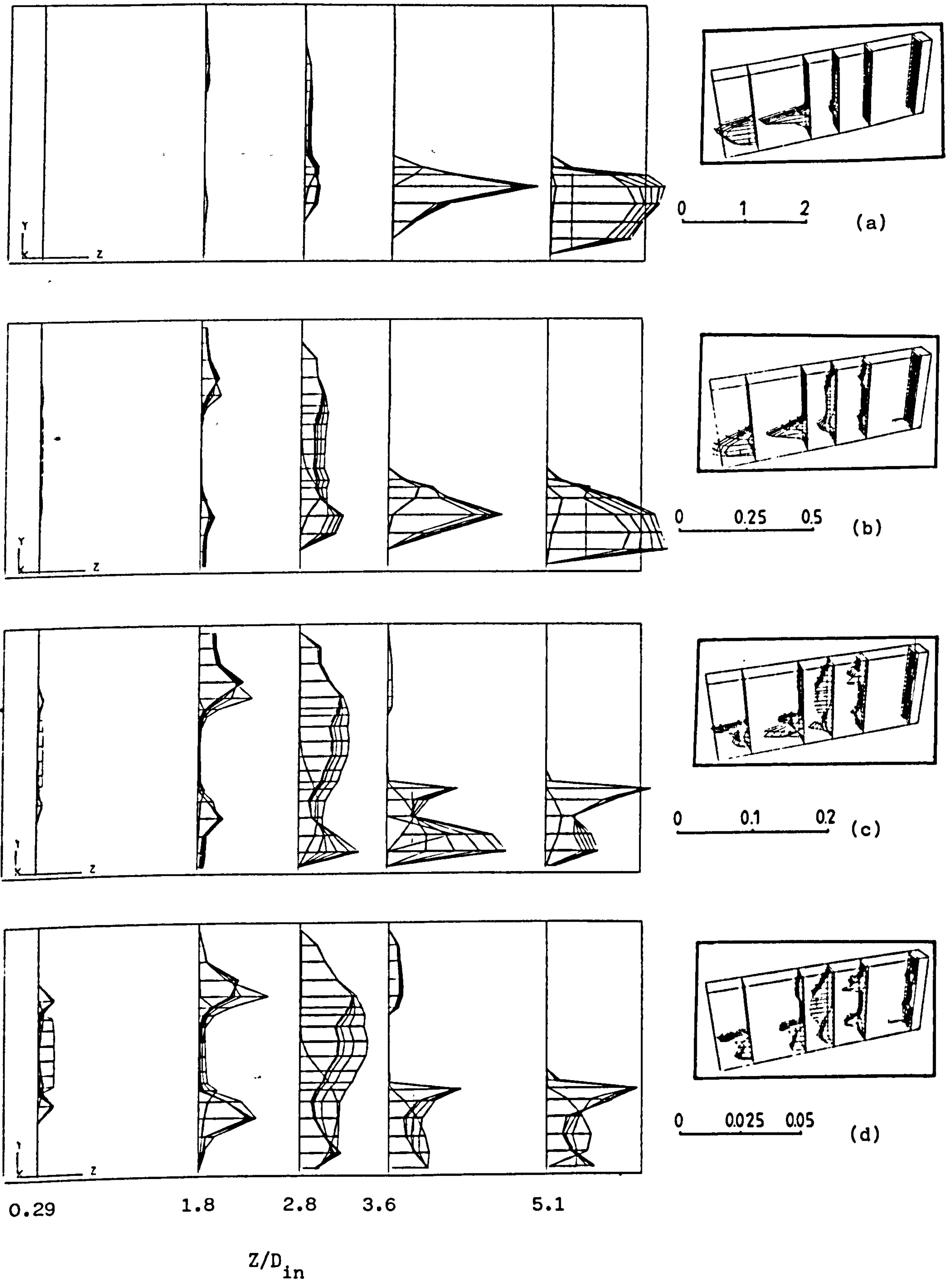


Fig. 6.12 Distribution of turbulence intensity ($I^2 = 2K/3 / U_0^2$) for the ramp throttle at 110° (a), 130° (b), 150° (c) and 180° (d).

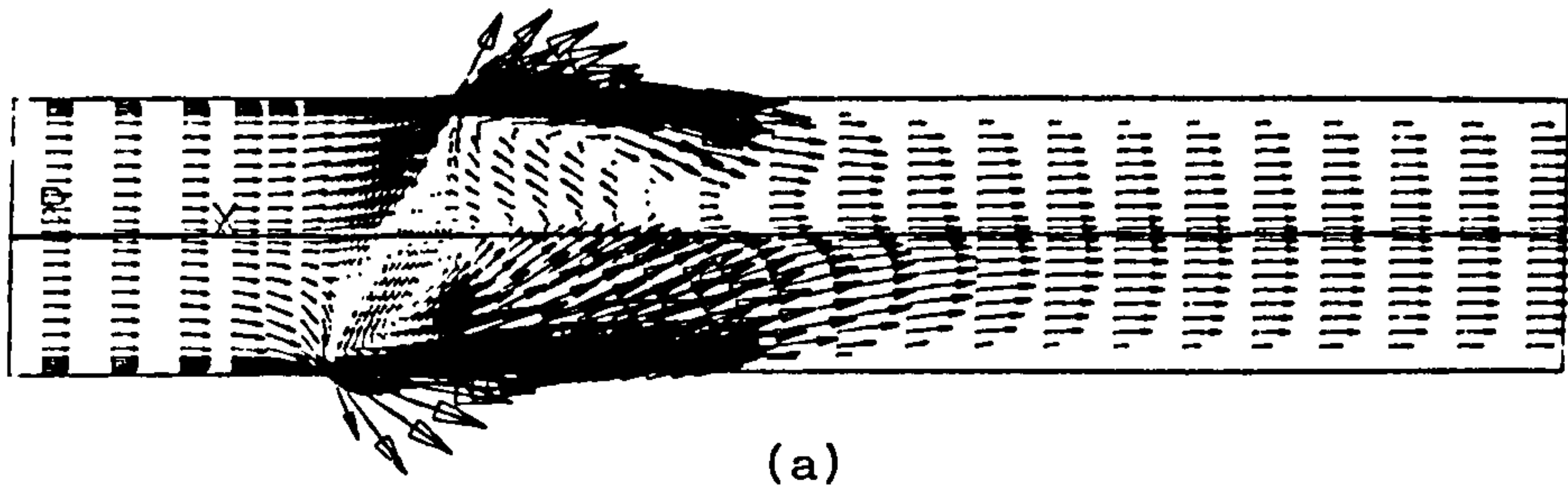
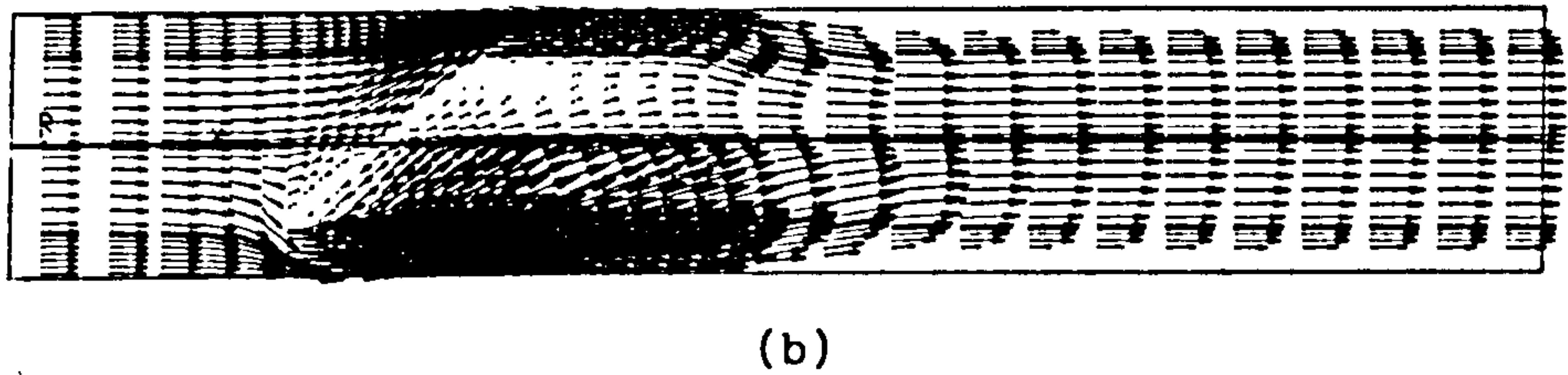
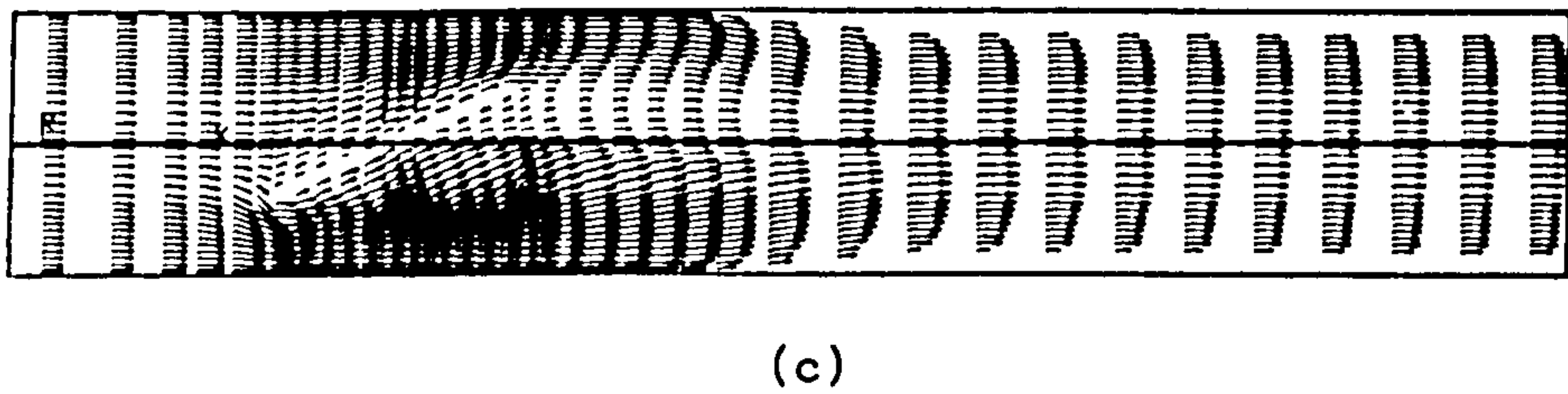
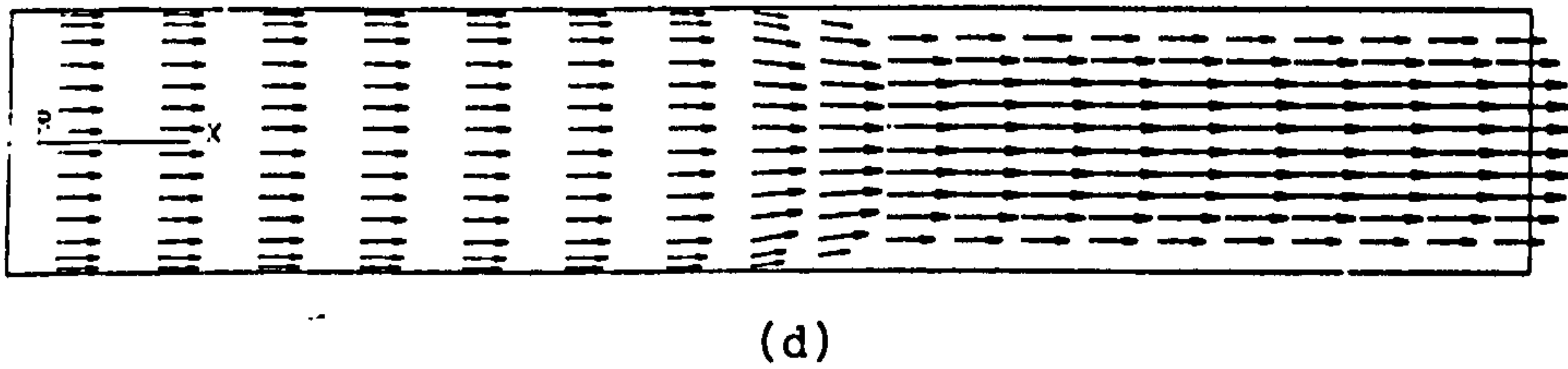


Fig. 6.13 Distribution of velocity vectors for the butterfly valve from computer simulation at settings of 30° (a), 50° (b), 65° (c) and 90° or wide open throttle (d).

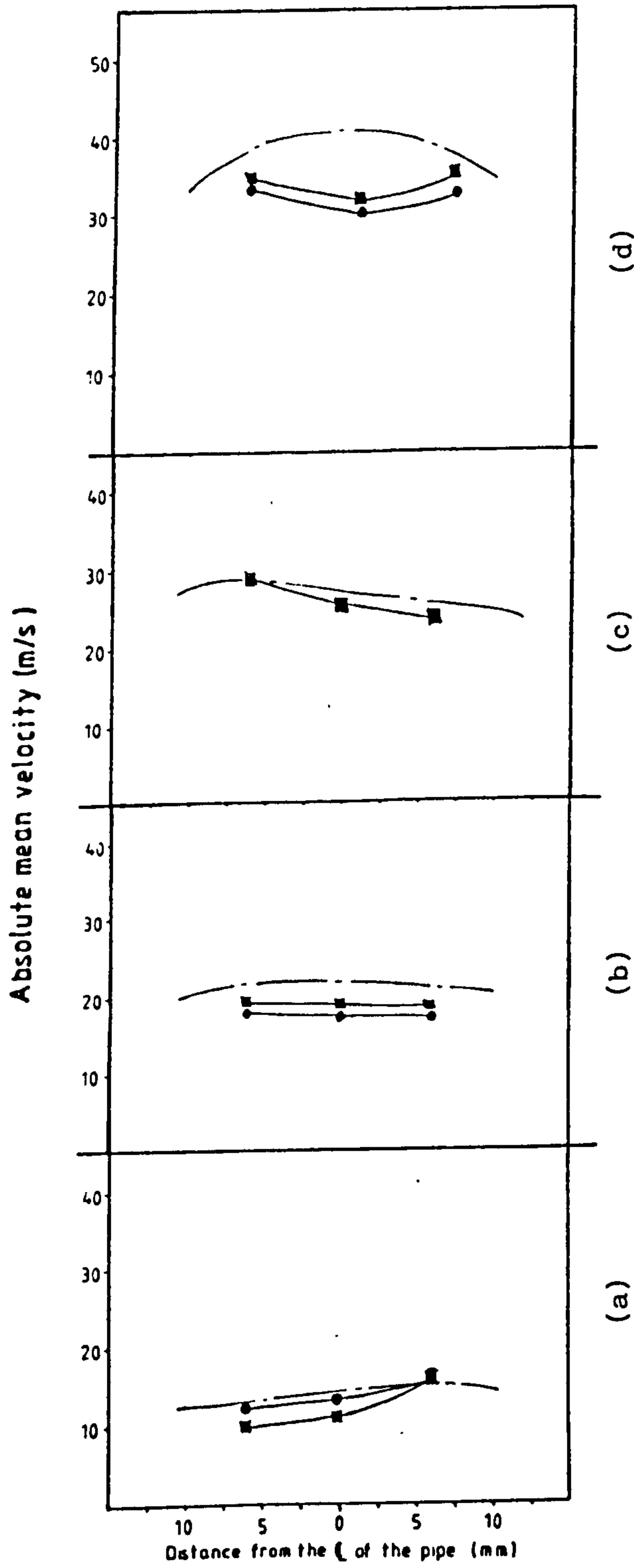
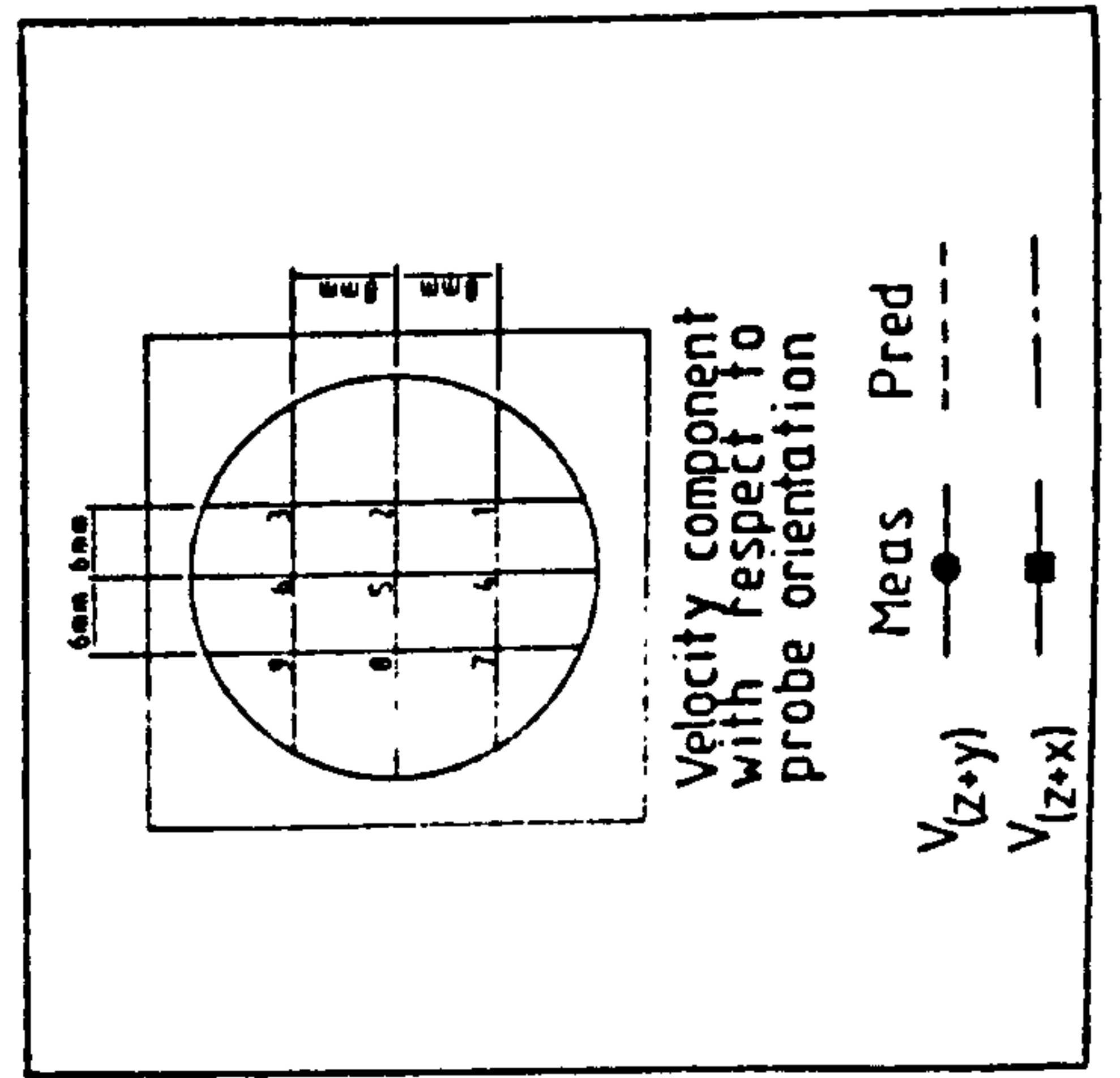


Fig. 6.14 Distribution of absolute mean velocities at plane (1-4-7) for butterfly valve angles of 30° (a), 50° (b), 65° (c) and 90° or wide open throttle (d).



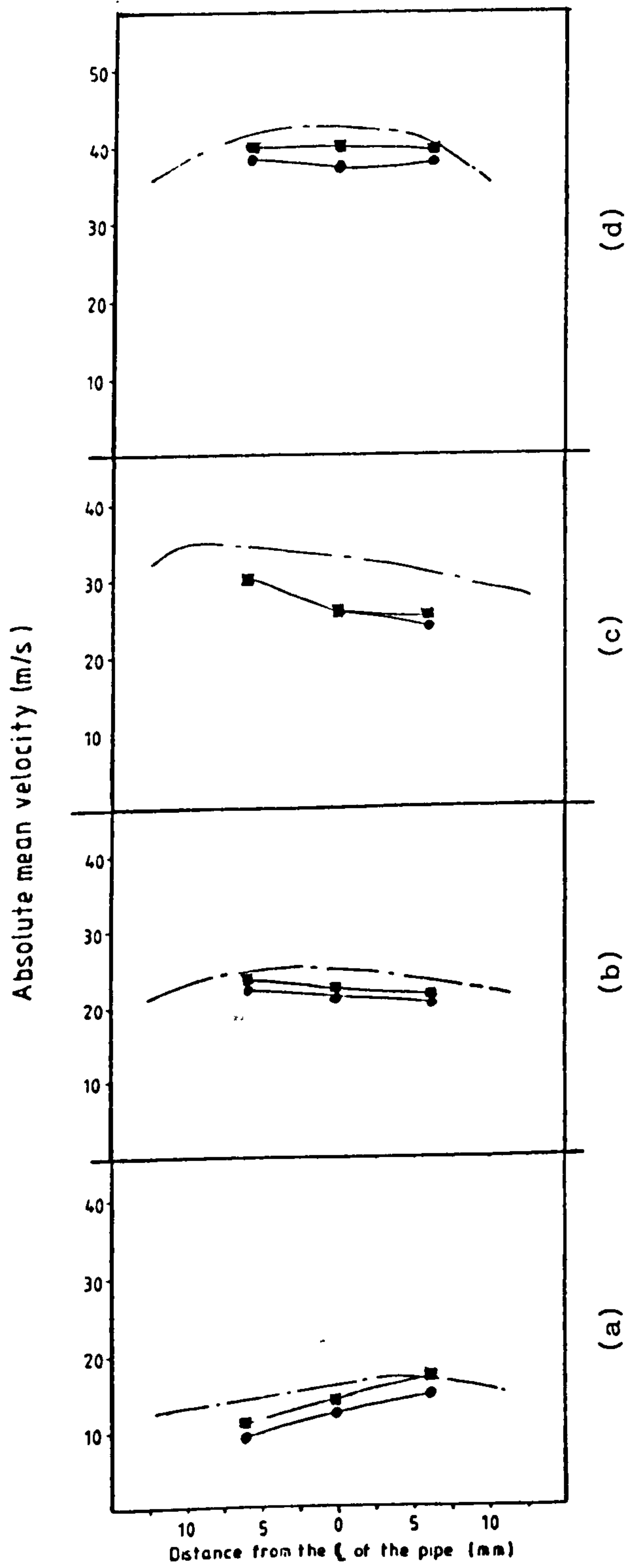
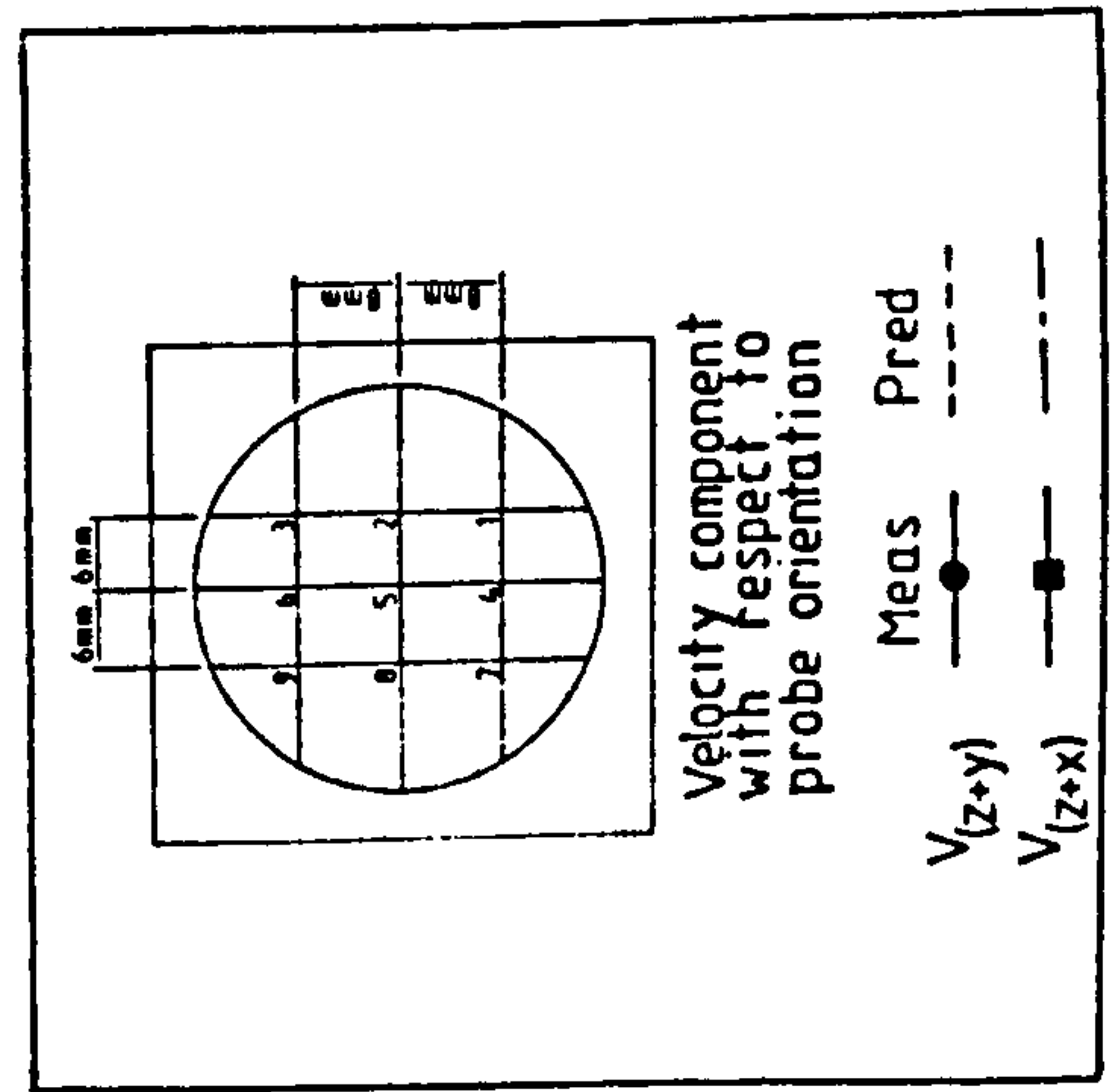


Fig. 6.15 Distribution of absolute mean velocities at plane (2-5-8) for butterfly valve angles of 30° (a), 50° (b), 65° (c) and 90° or wide open throttle (d).



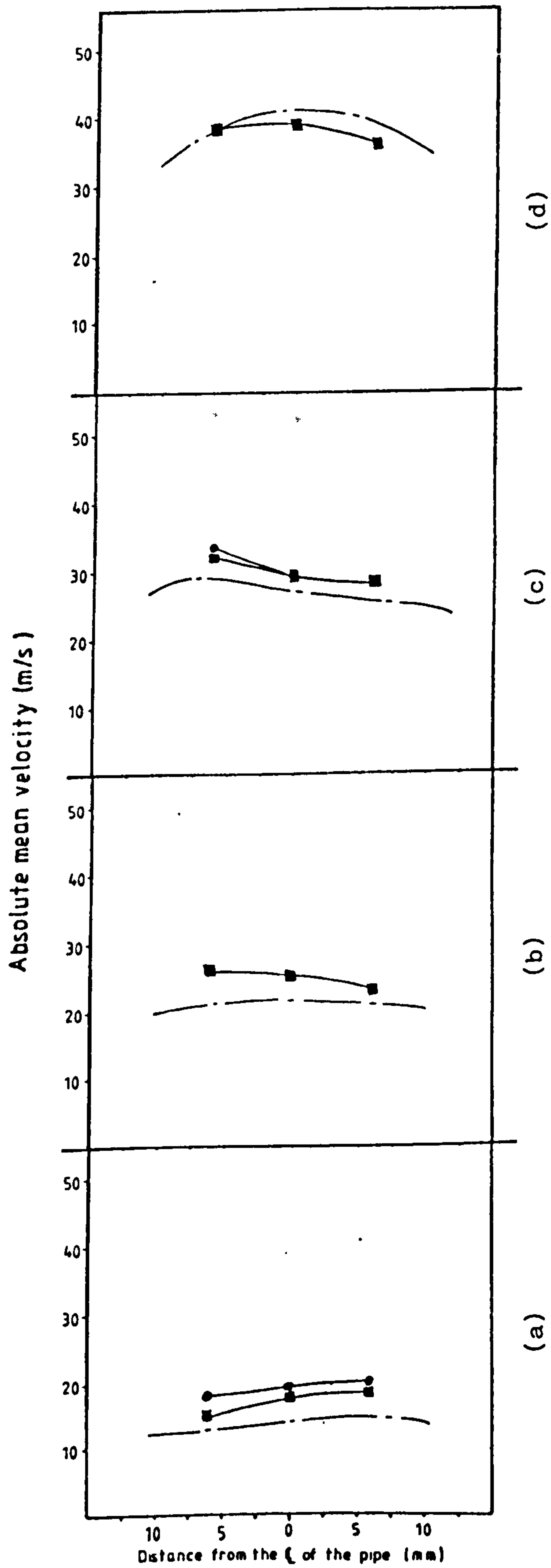
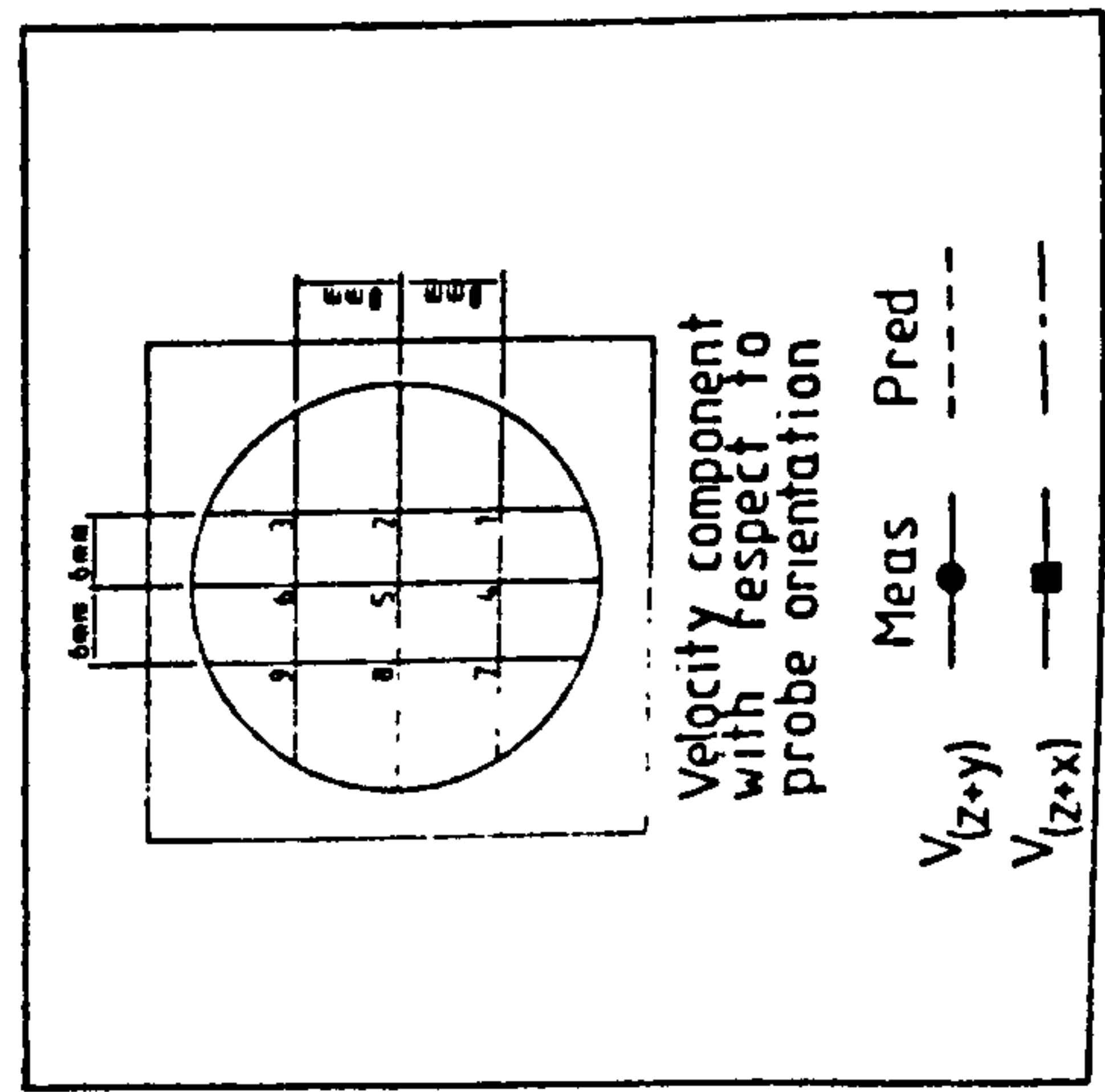


Fig. 6.16 Distribution of absolute mean velocities at plane (3-6-9) for butterfly valve angles of 30° (a), 50° (b), 65° (c) and 90° or wide open throttle (d).



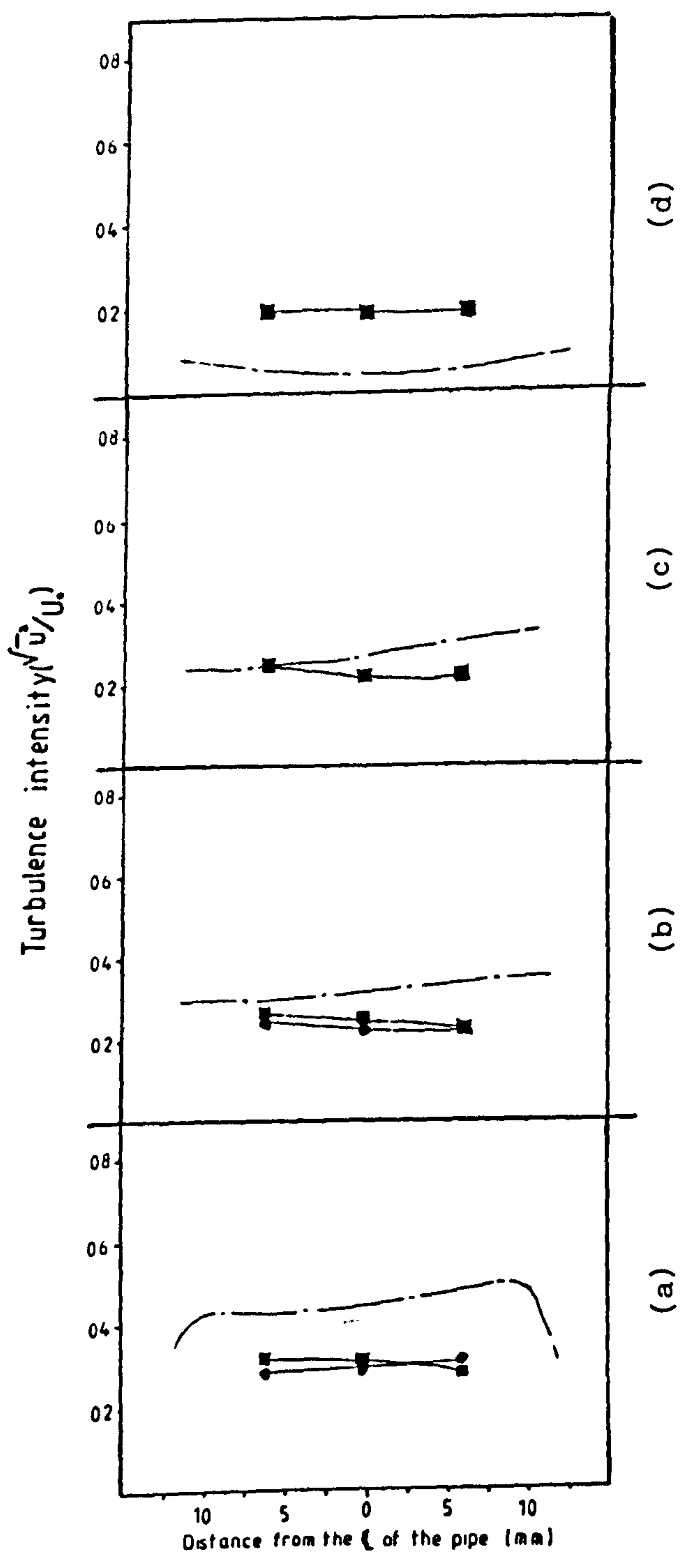
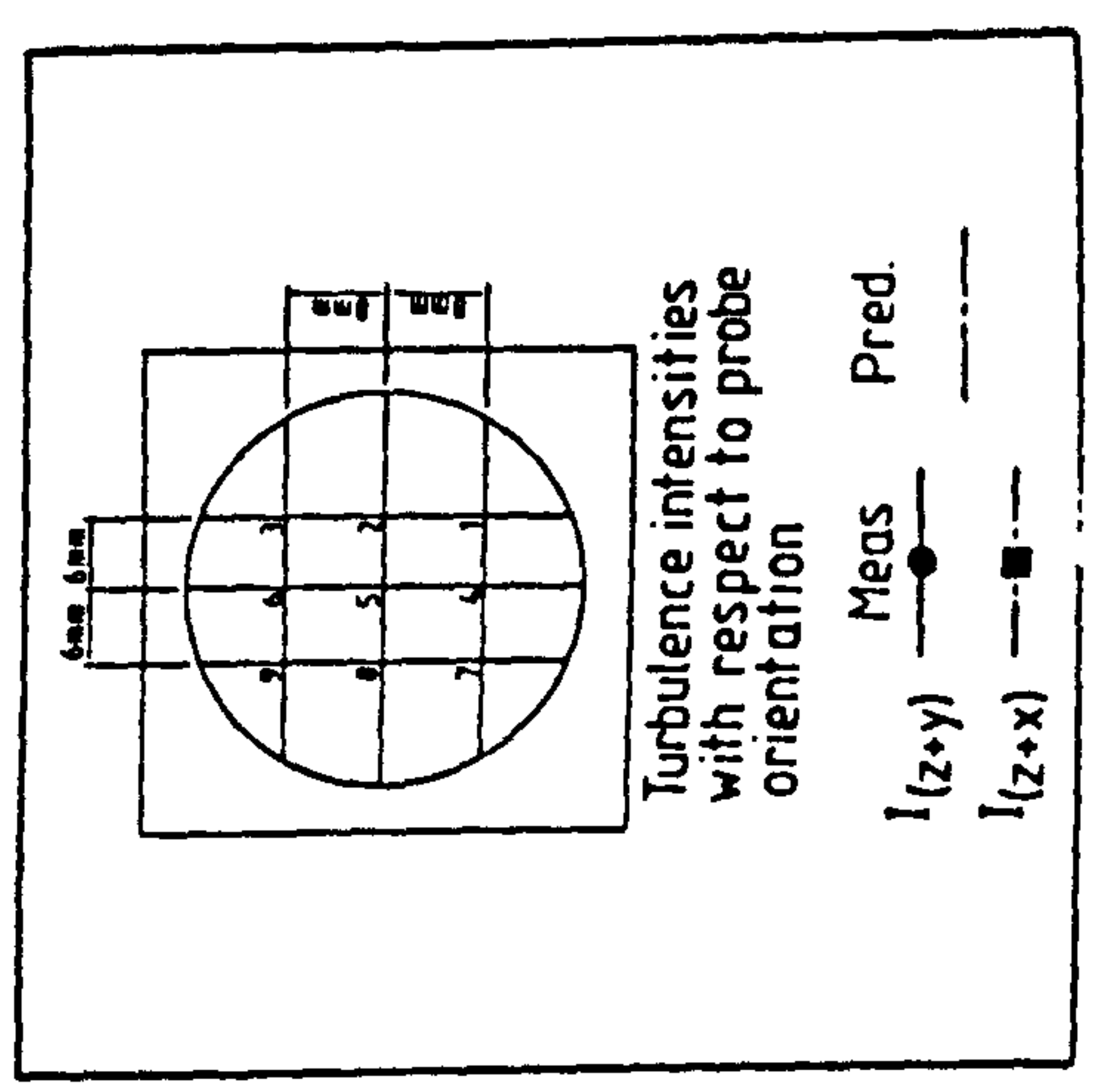


Fig. 6.17 Distribution of turbulence intensity ($u^2 = \frac{2}{3} K$) at plane (1-4-7) for butterfly valve angles of 30° (a), 50° (b), 65° (c) and 90° (d).



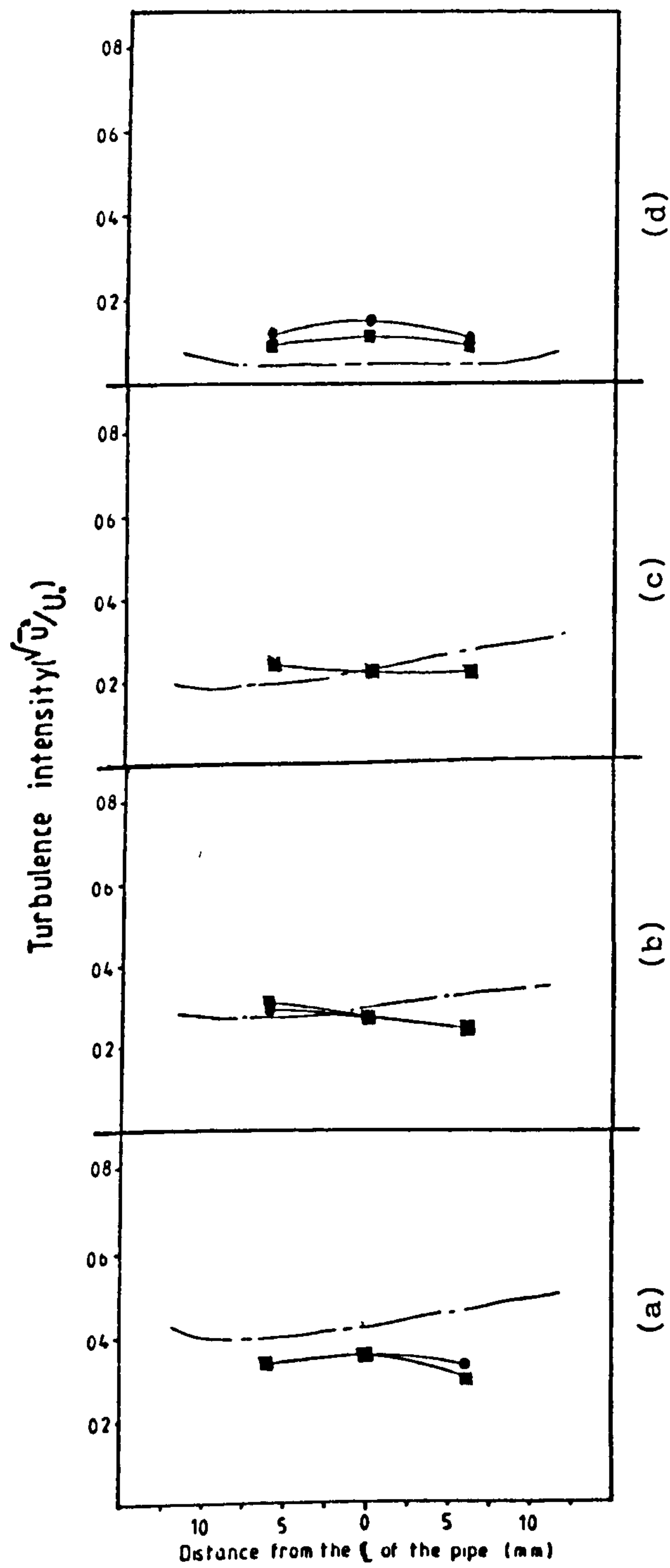
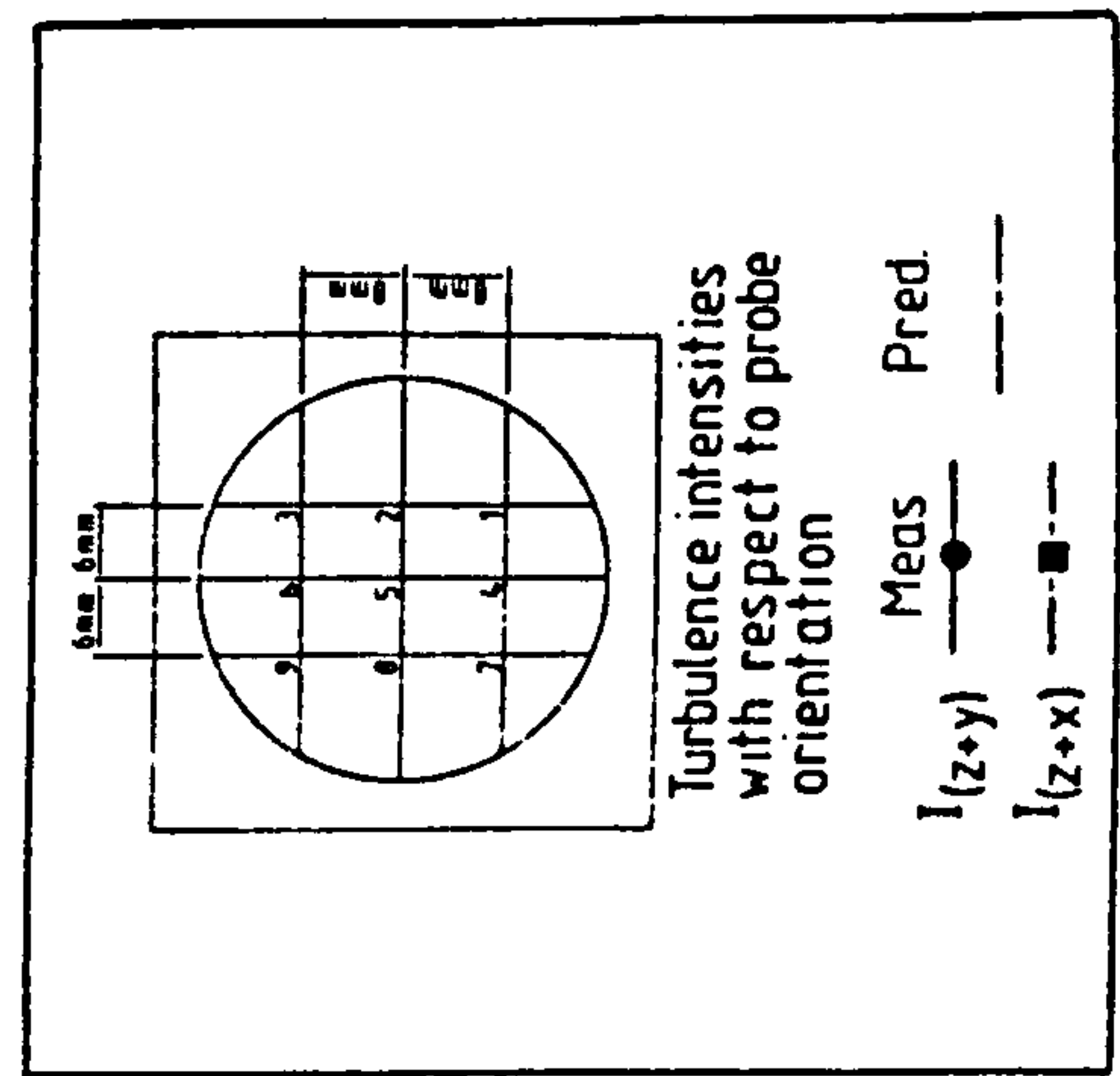


Fig. 6.18 Distribution of turbulence intensity ($u^2 = \frac{2}{3} K$) at plane (2-5-8) for butterfly valve angles of 30° (a), 50° (b), 65° (c) and 90° (d).



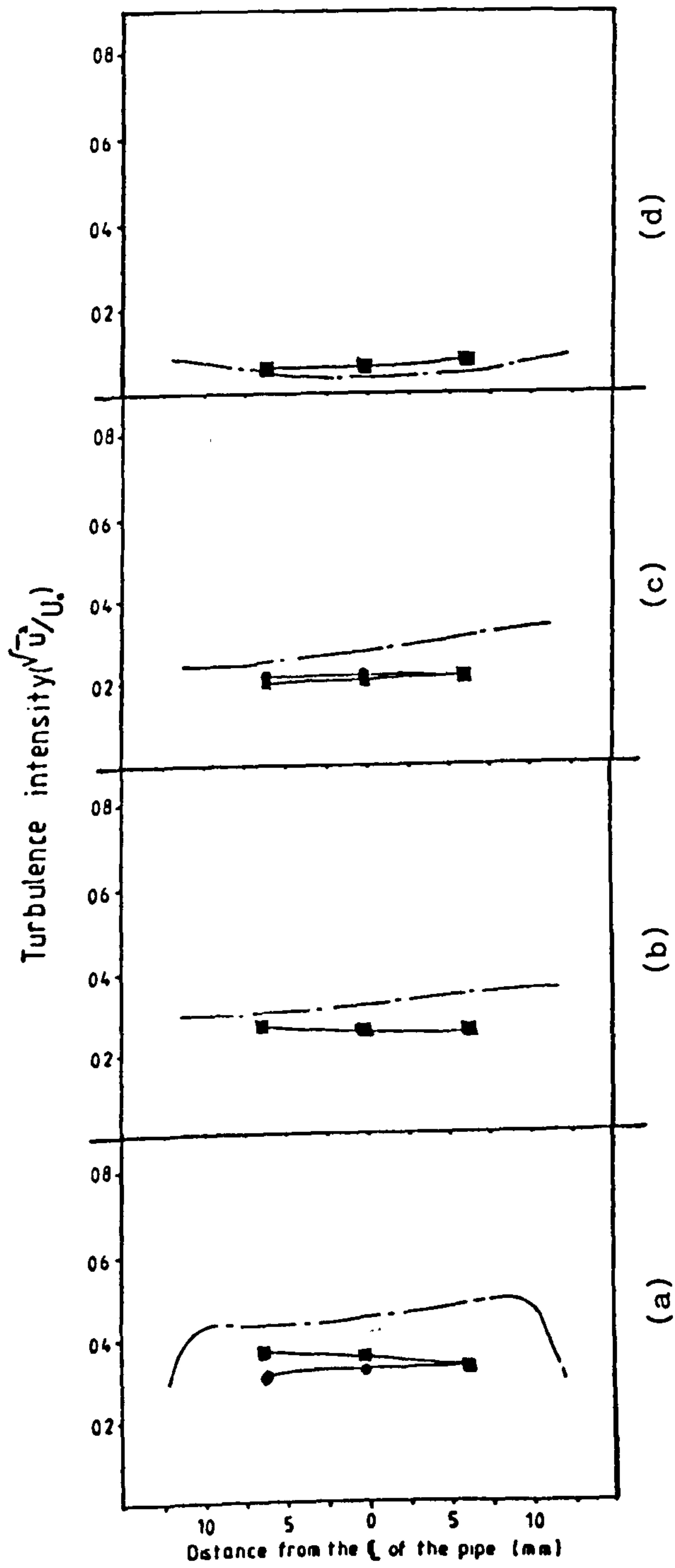
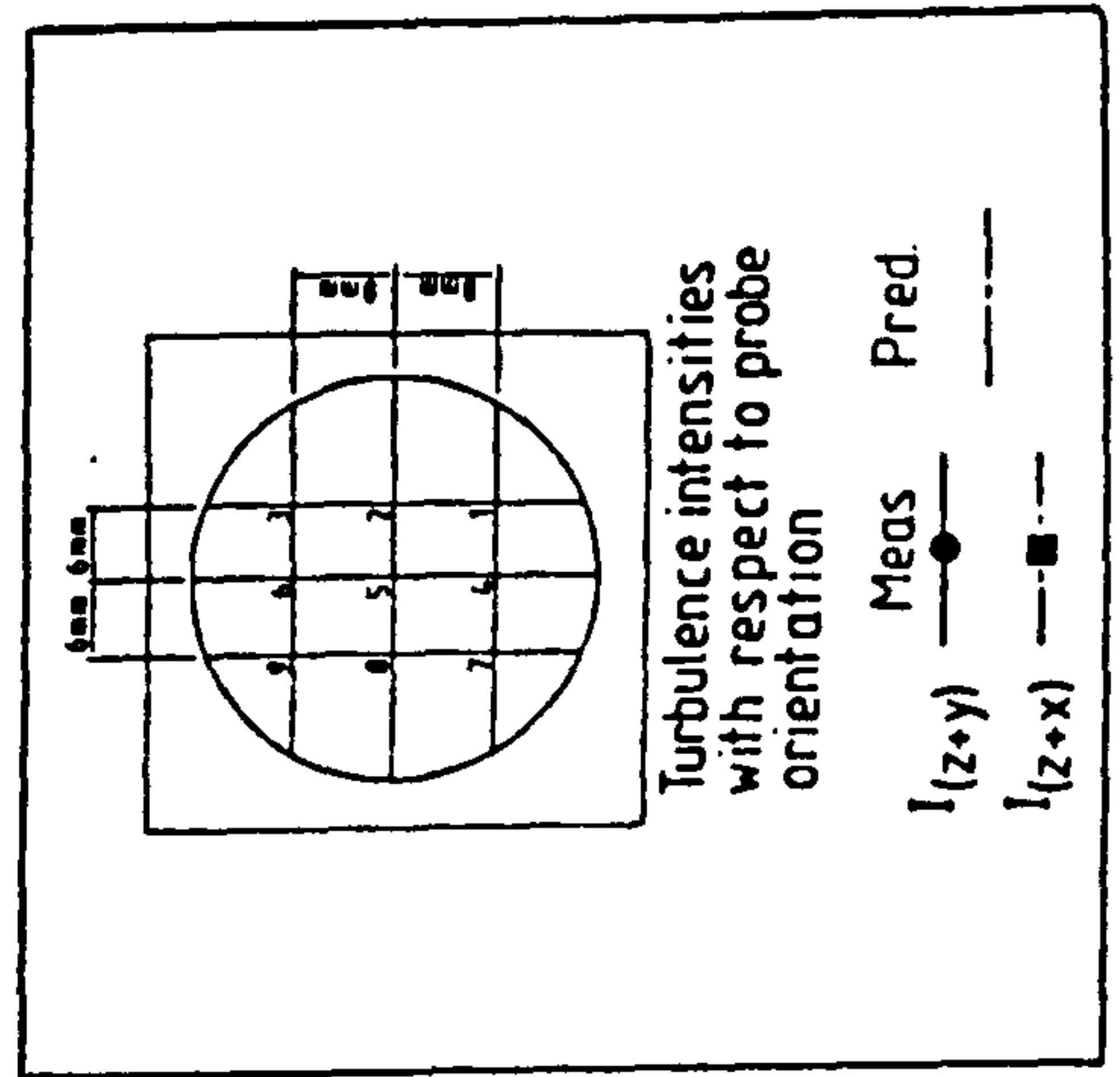


Fig. 6.19 Distribution of turbulence intensity ($u^2 = \frac{2}{3} K$) at plane (3-6-9) for butterfly valve angles of 30° (a), 50° (b), 65° (c) and 90° (d).



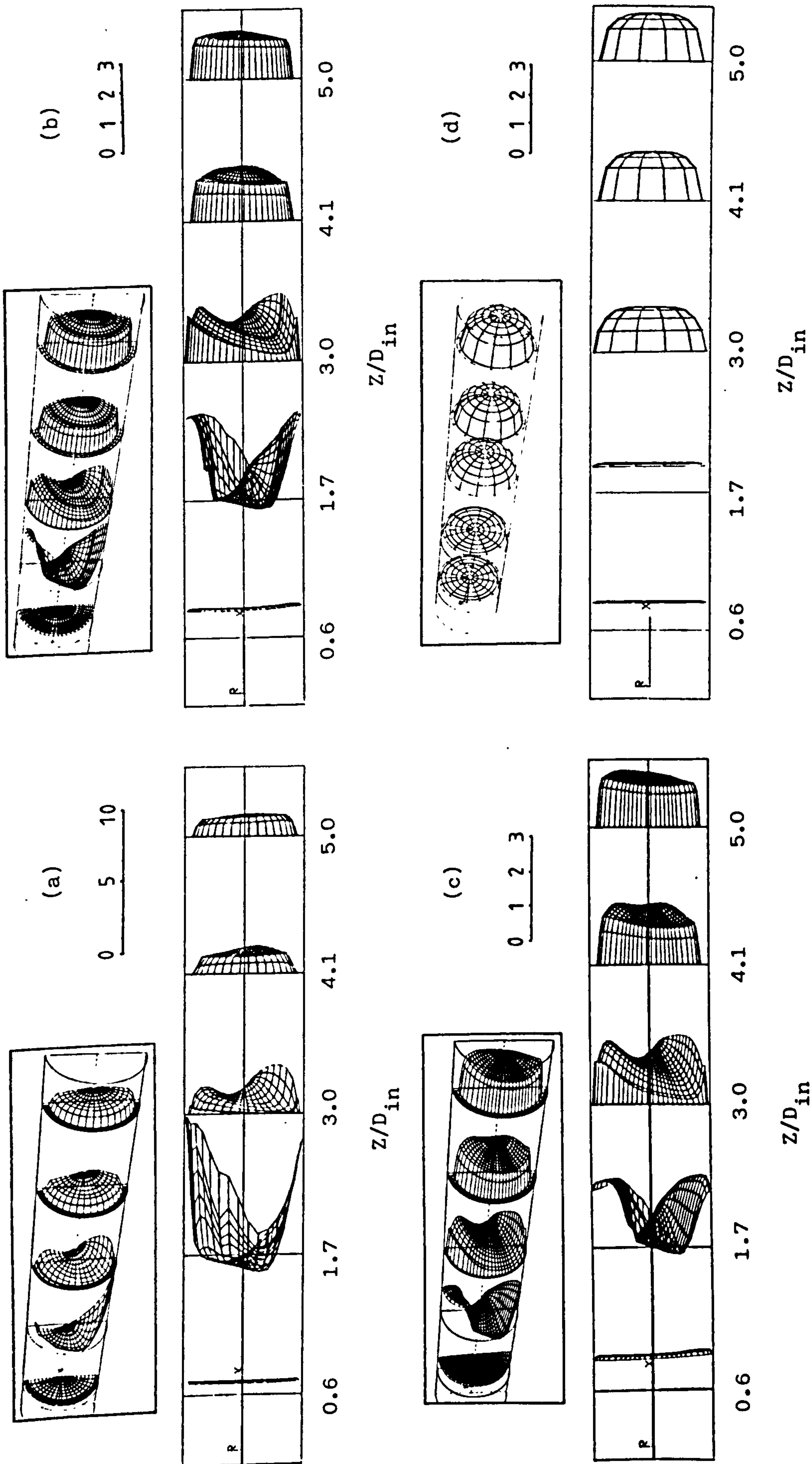


Fig. 6.20 Distribution of mean axial velocity (U_z/U_o) for butterfly valve angles of 30° (a), 50° (b), 65° (c) and 90° (d).

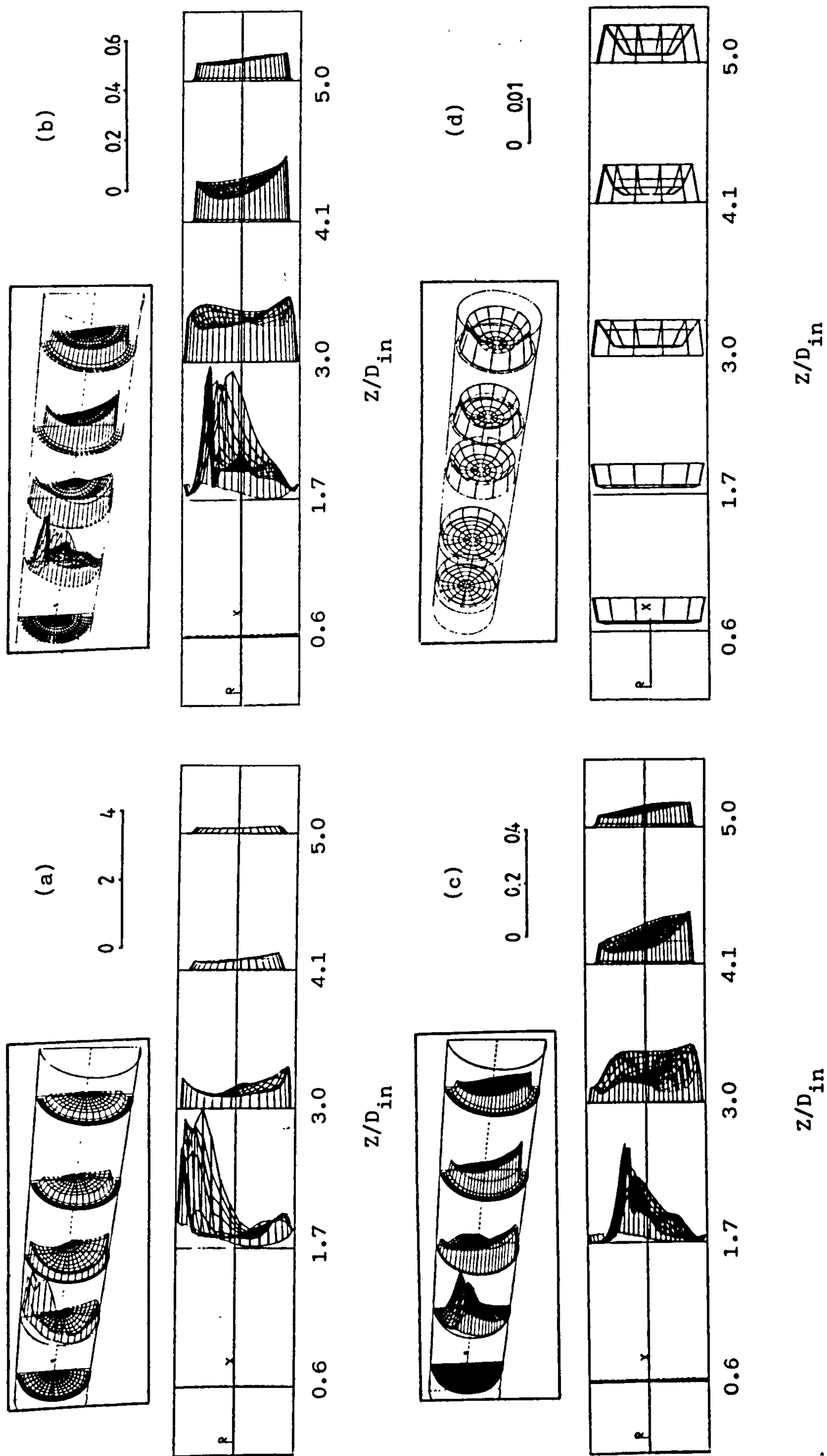


Fig. 6.21 Distribution of turbulence intensity ($I^2 = 2K/3 / U_0^2$) for butterfly valve angles of 30° (a), 50° (b), 65° (c) and 90° (d).

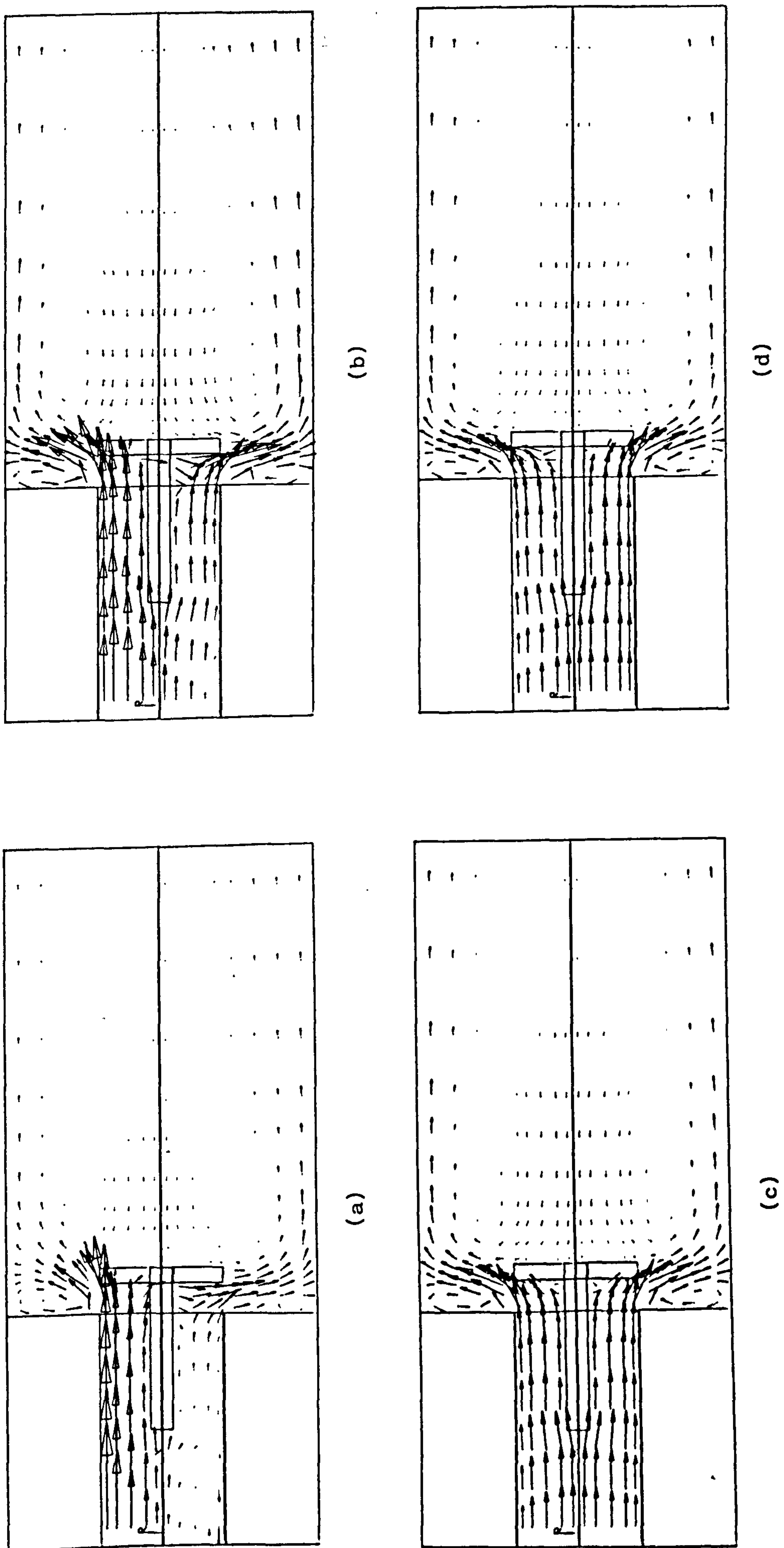


Fig. 6.22 Mid-plane distributions of velocity vectors on air admission into cylinder for ramp throttle included angles of 110° (a), 130° (b), 150° (c) and 180° (d).

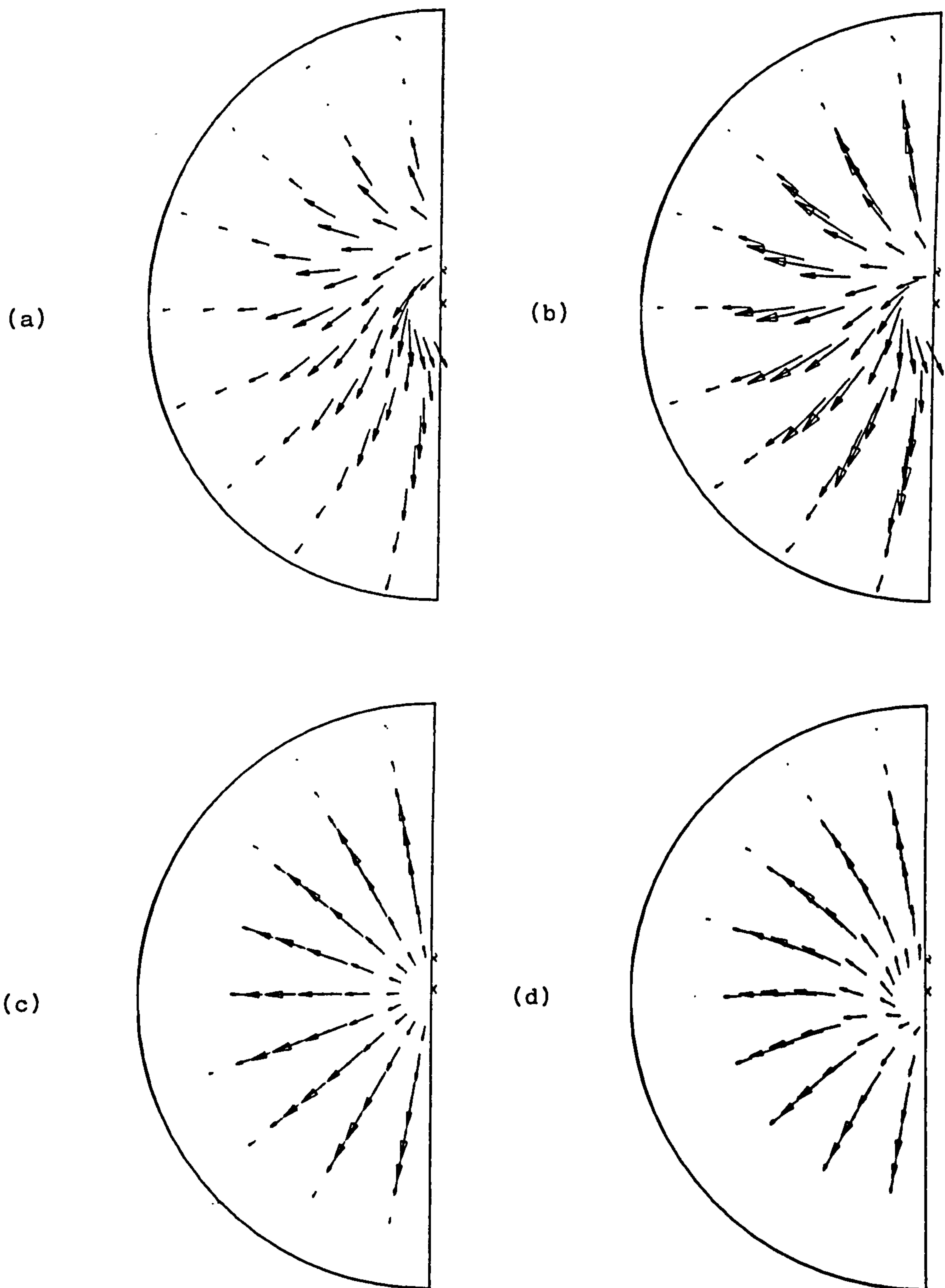
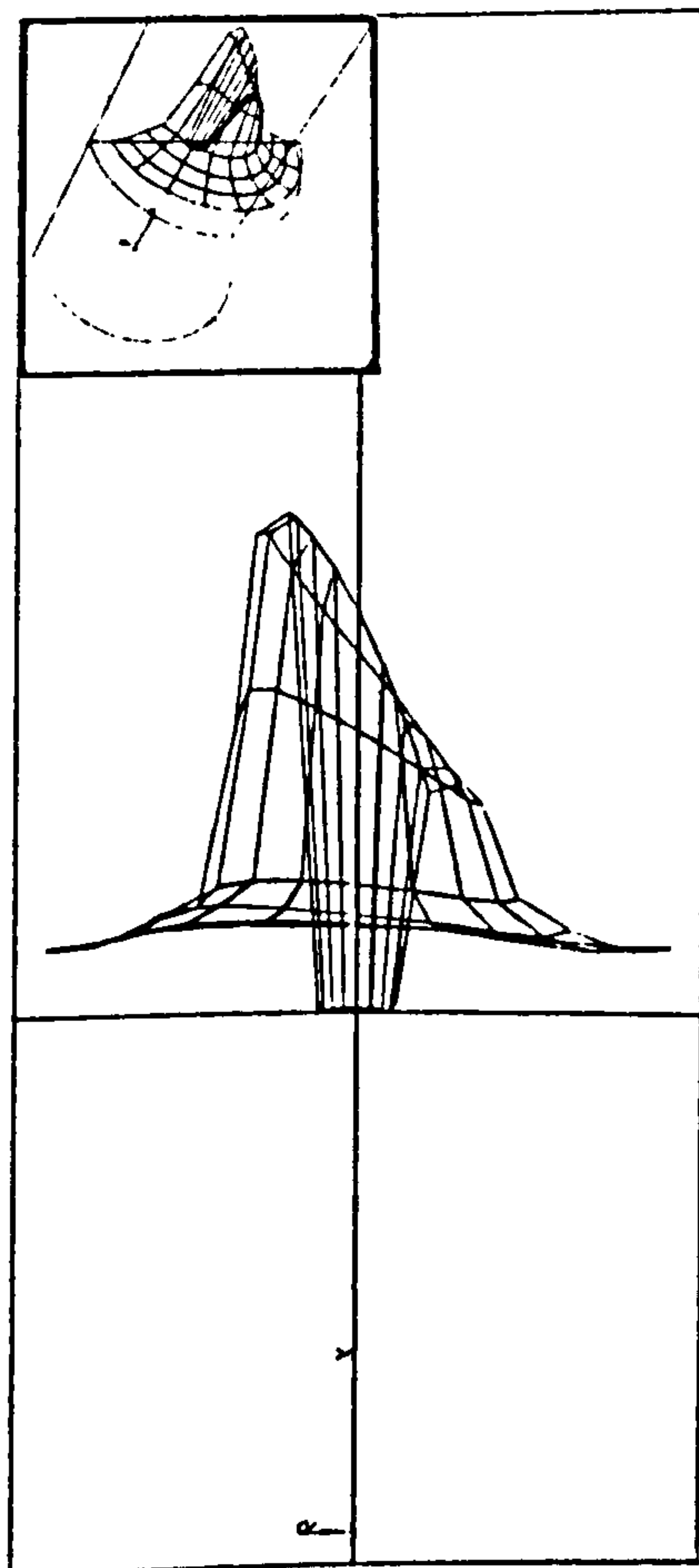
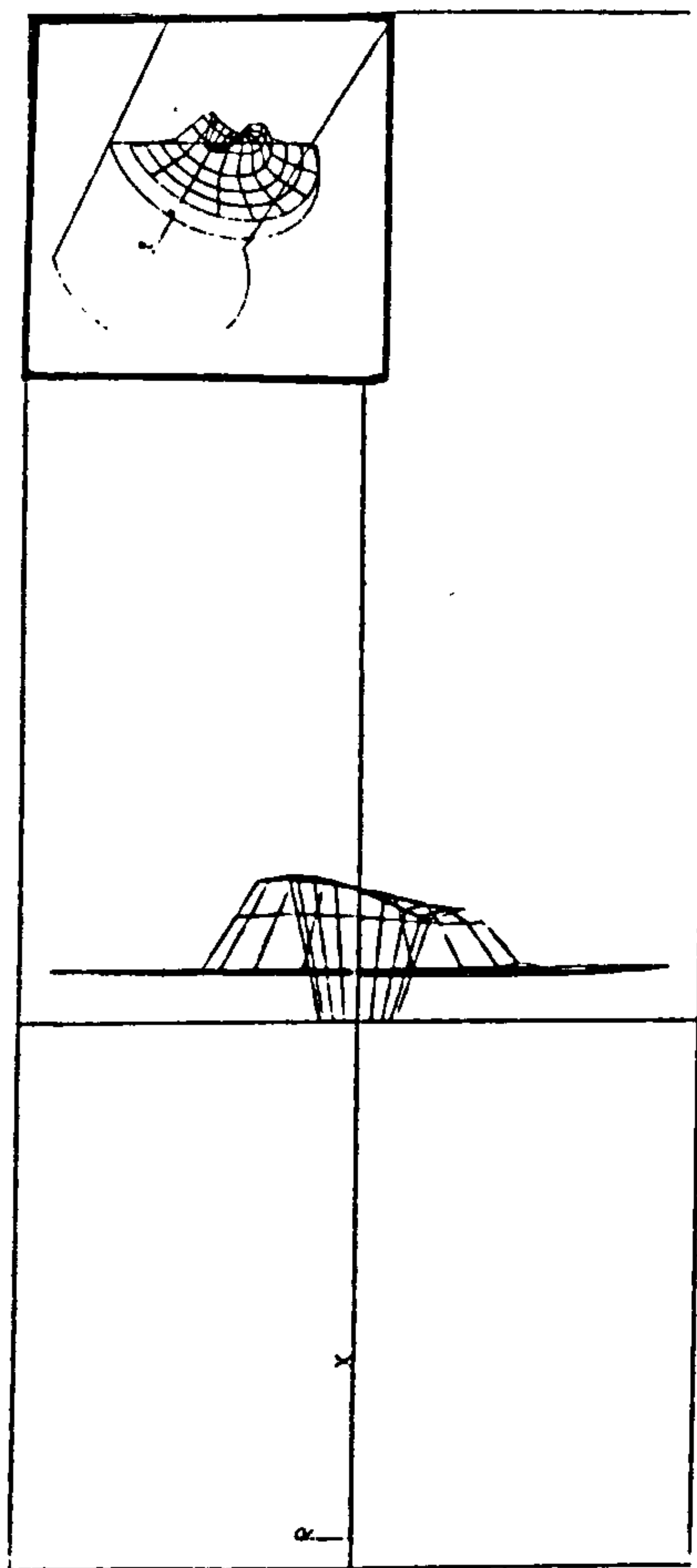


Fig. 6.23 Distribution of velocity vectors in the traverse plane of the inlet valve for ramp throttle at included angles of 110° (a), 130° (b), 150° (c) and 180° (d).

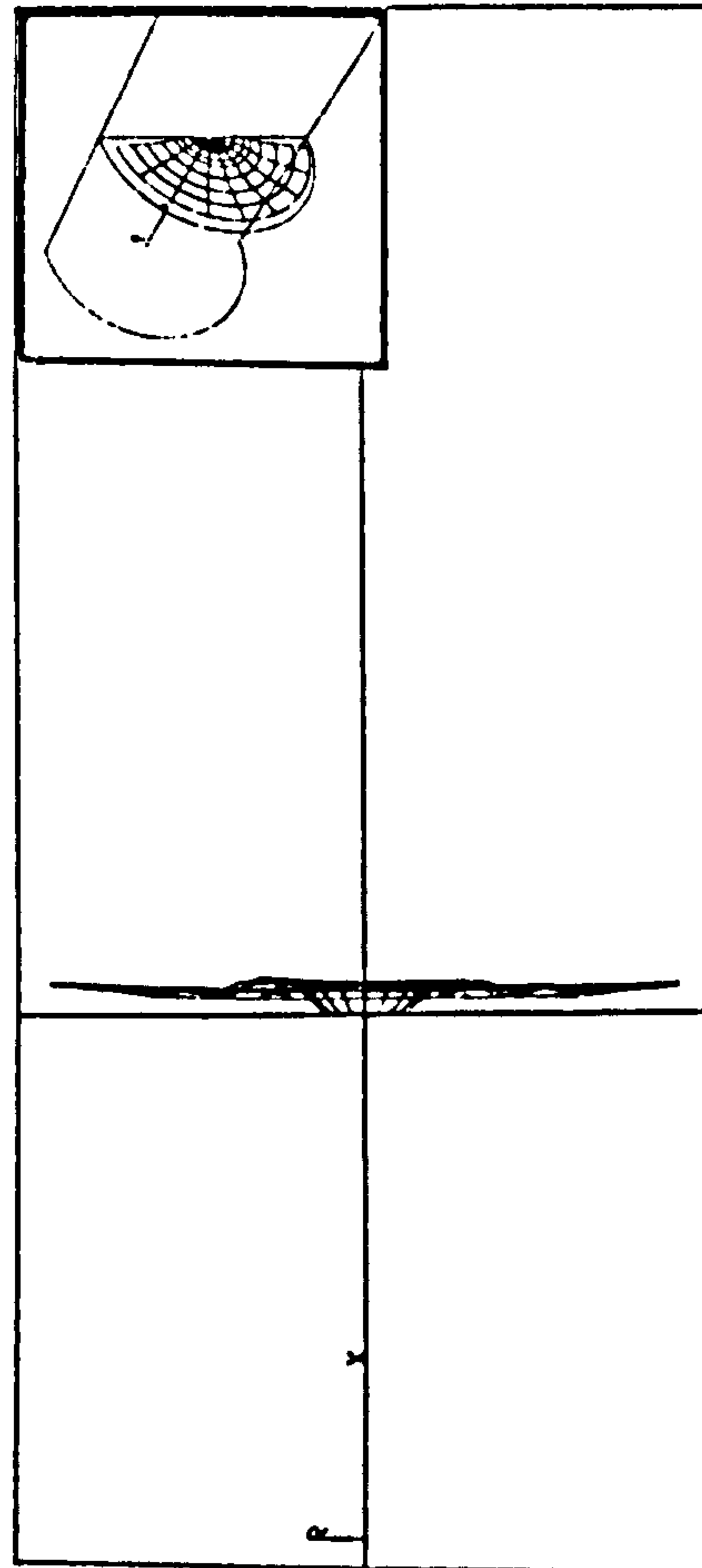
0 1 2



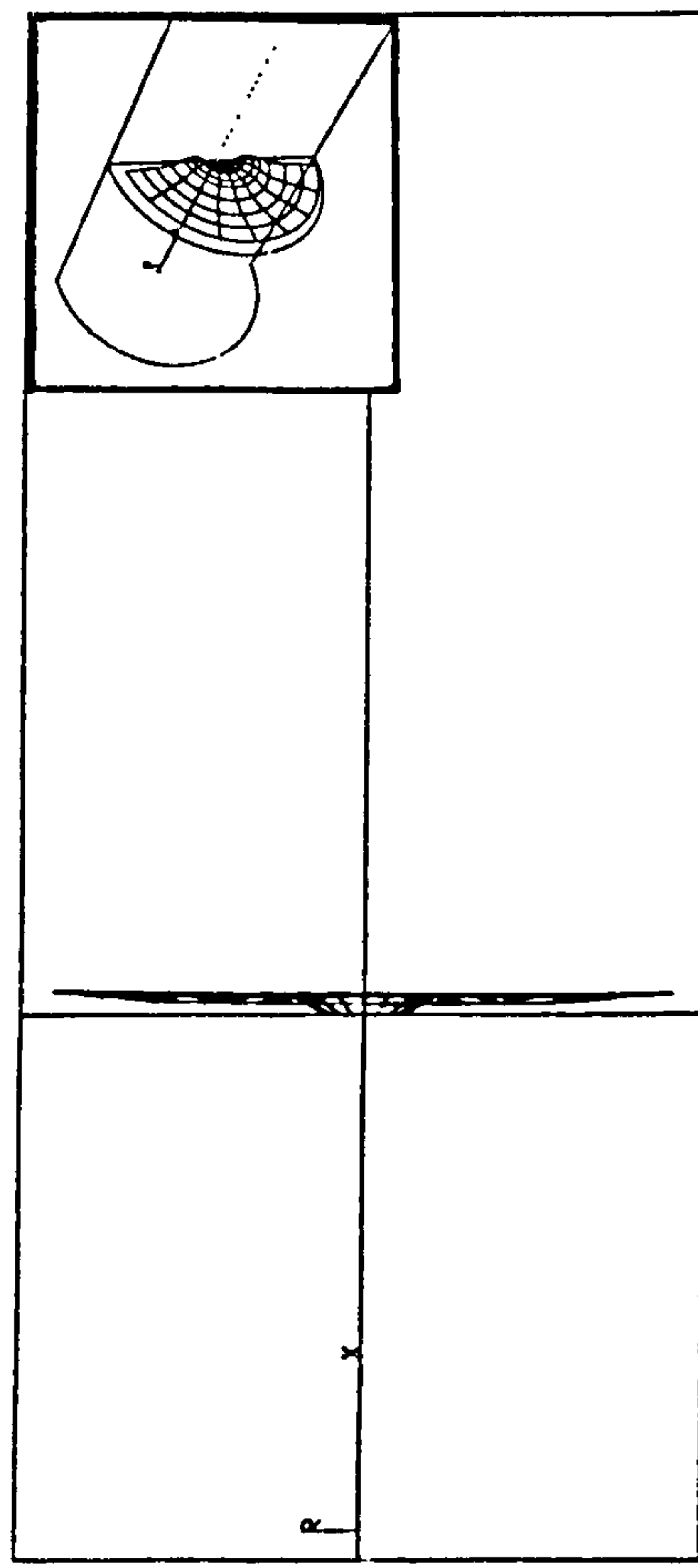
(a)



(b)

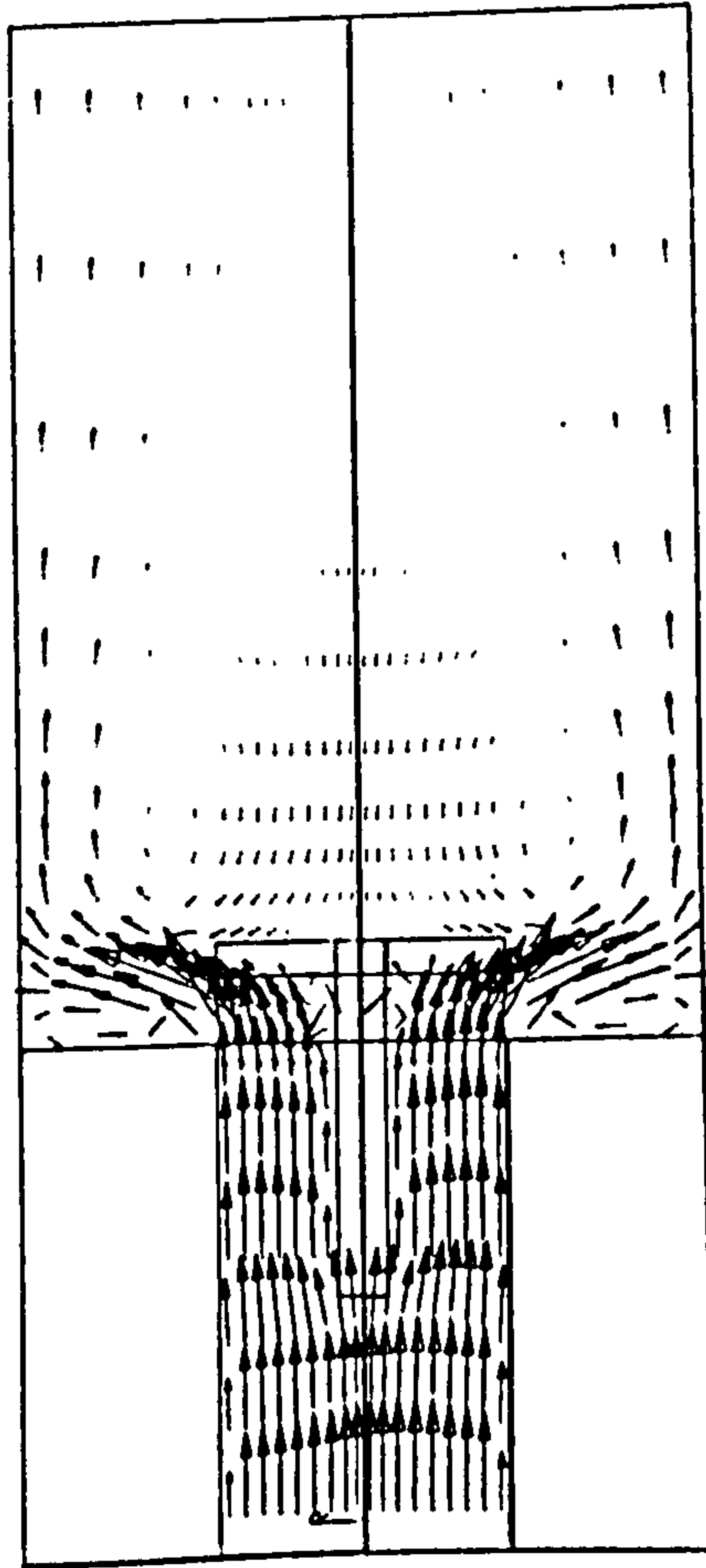


(c)

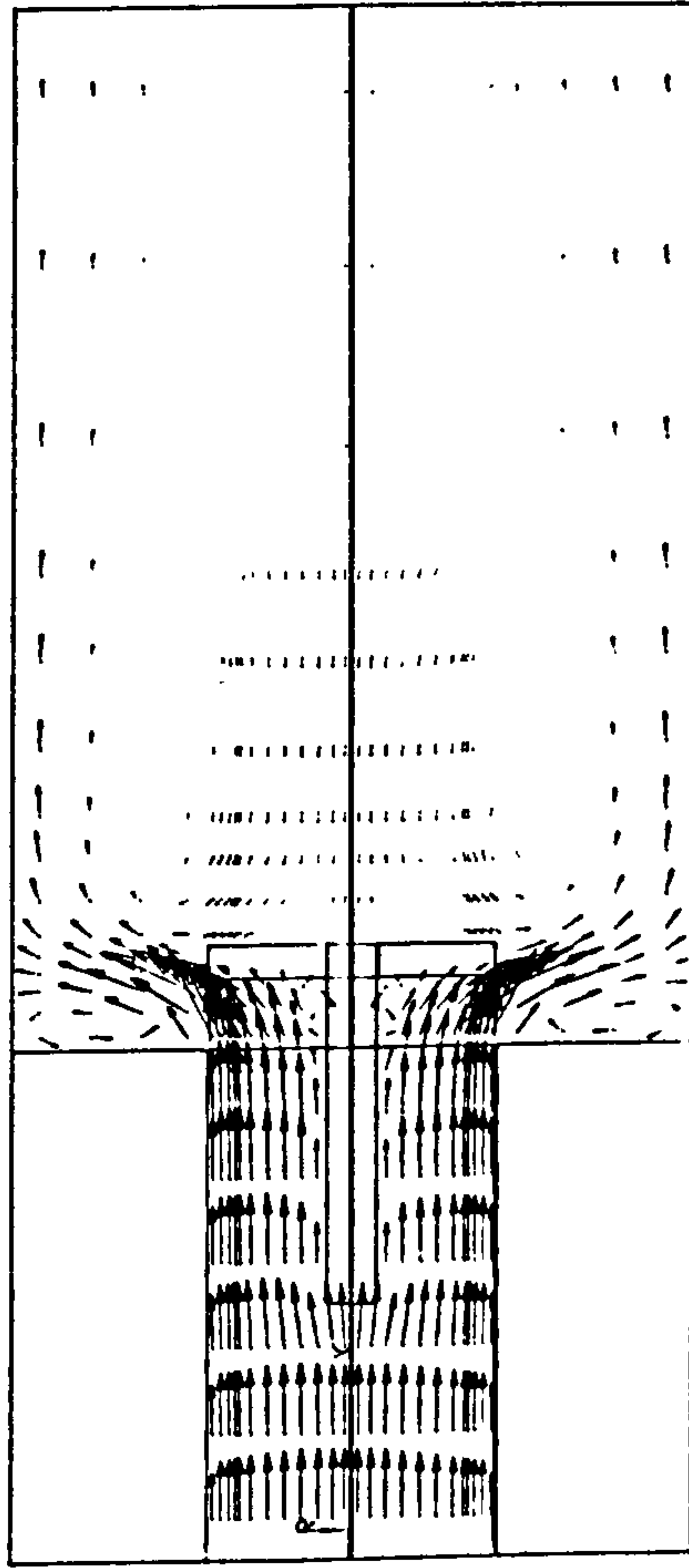


(d)

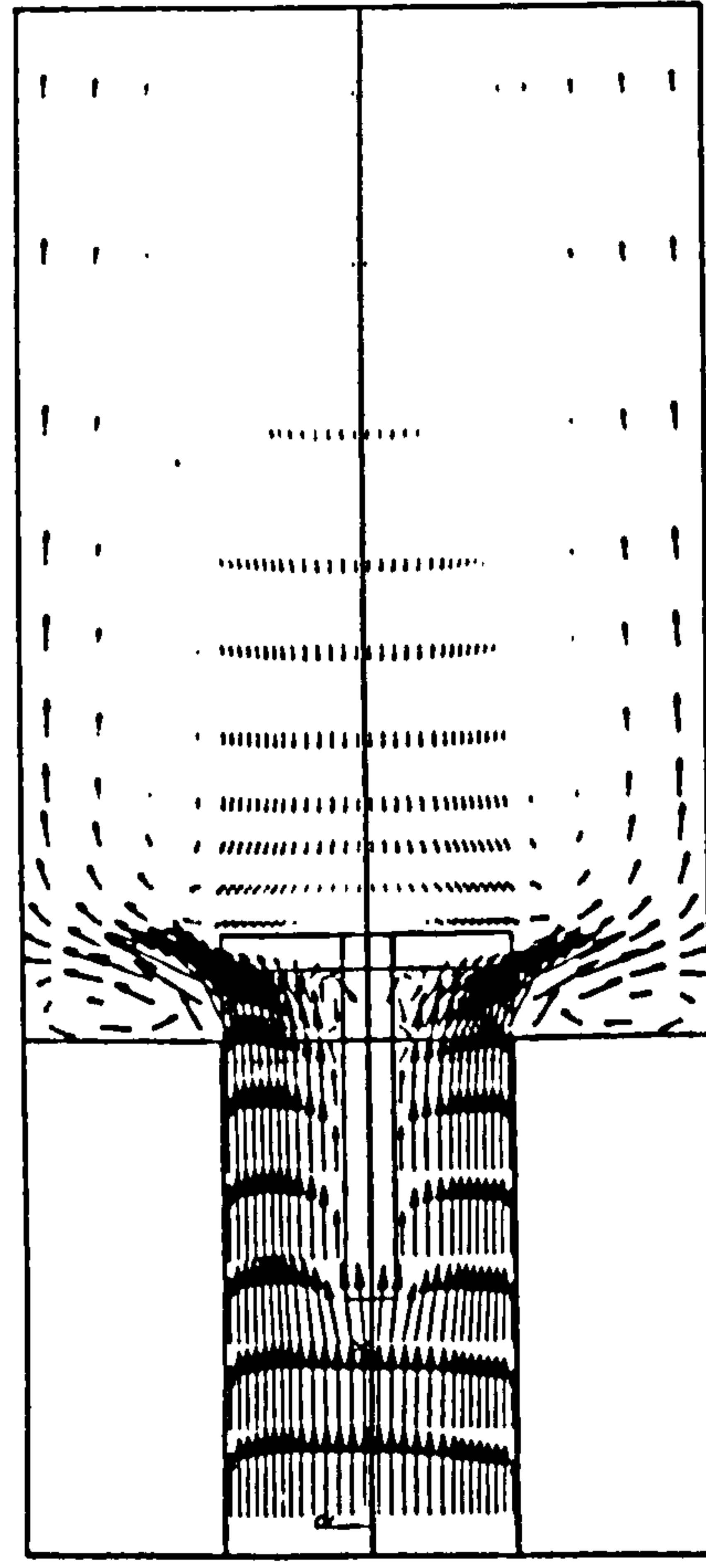
Fig. 6.24 Distribution of turbulence intensity ($I^2 = \frac{2}{3} K / U^2$) at the cylinder head for ramp throttle included angles of 110° (a), 130° (b), 150° (c) and 180° (d).



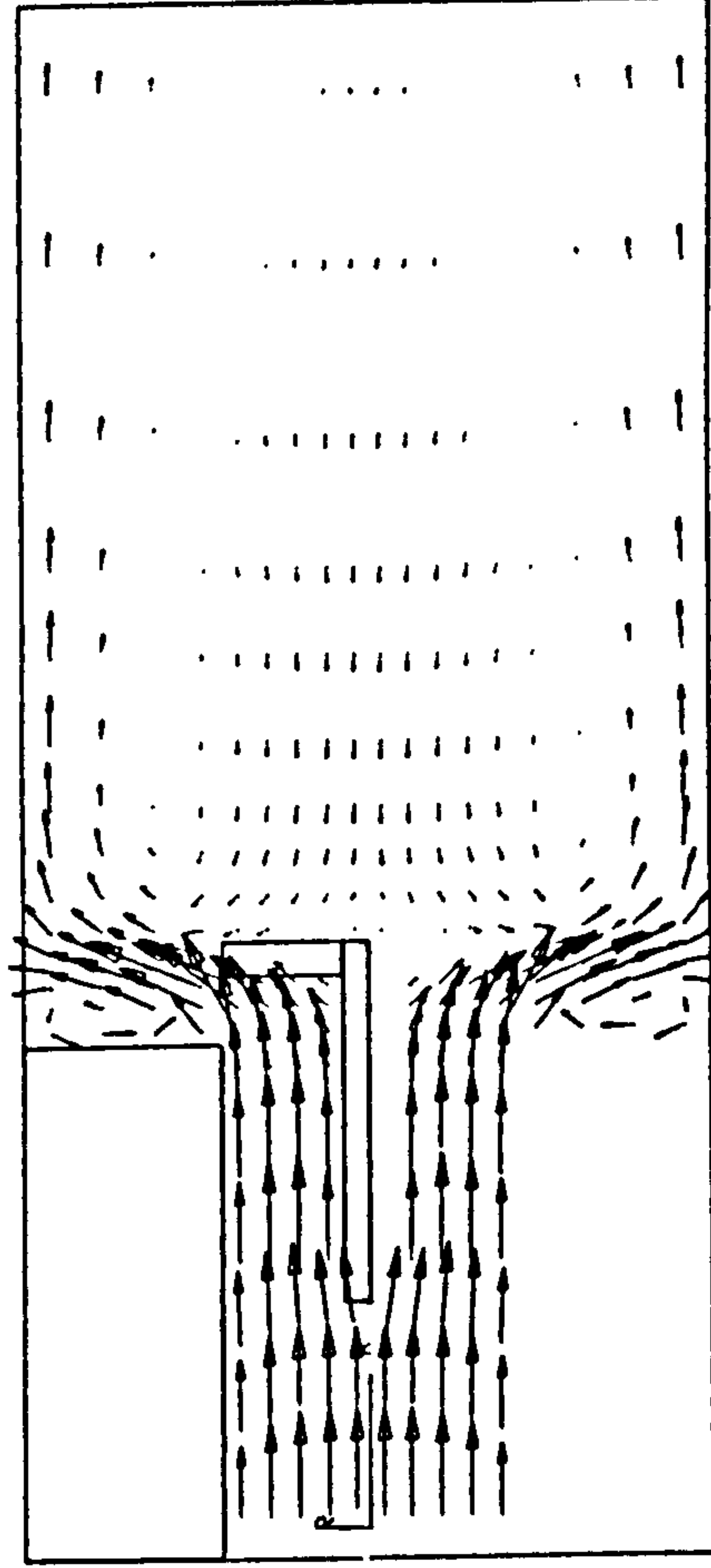
(a)



(b)



(c)



(d)

Fig. 6.25 Mid-plane distribution of velocity vectors on air admission into the cylinder for butterfly valve angles of 30° (a), 50° (b), 65° (c) and 90° (d).

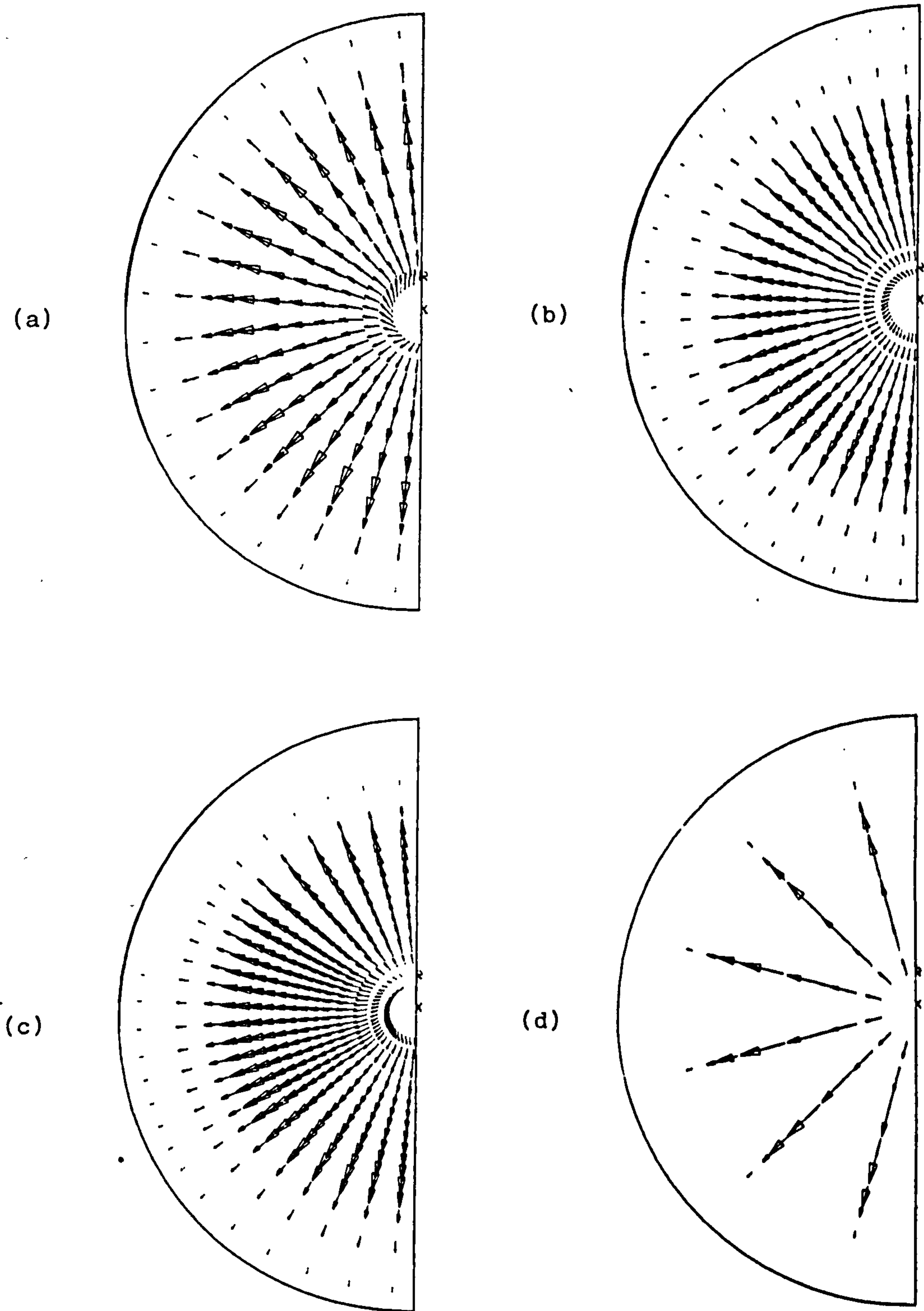
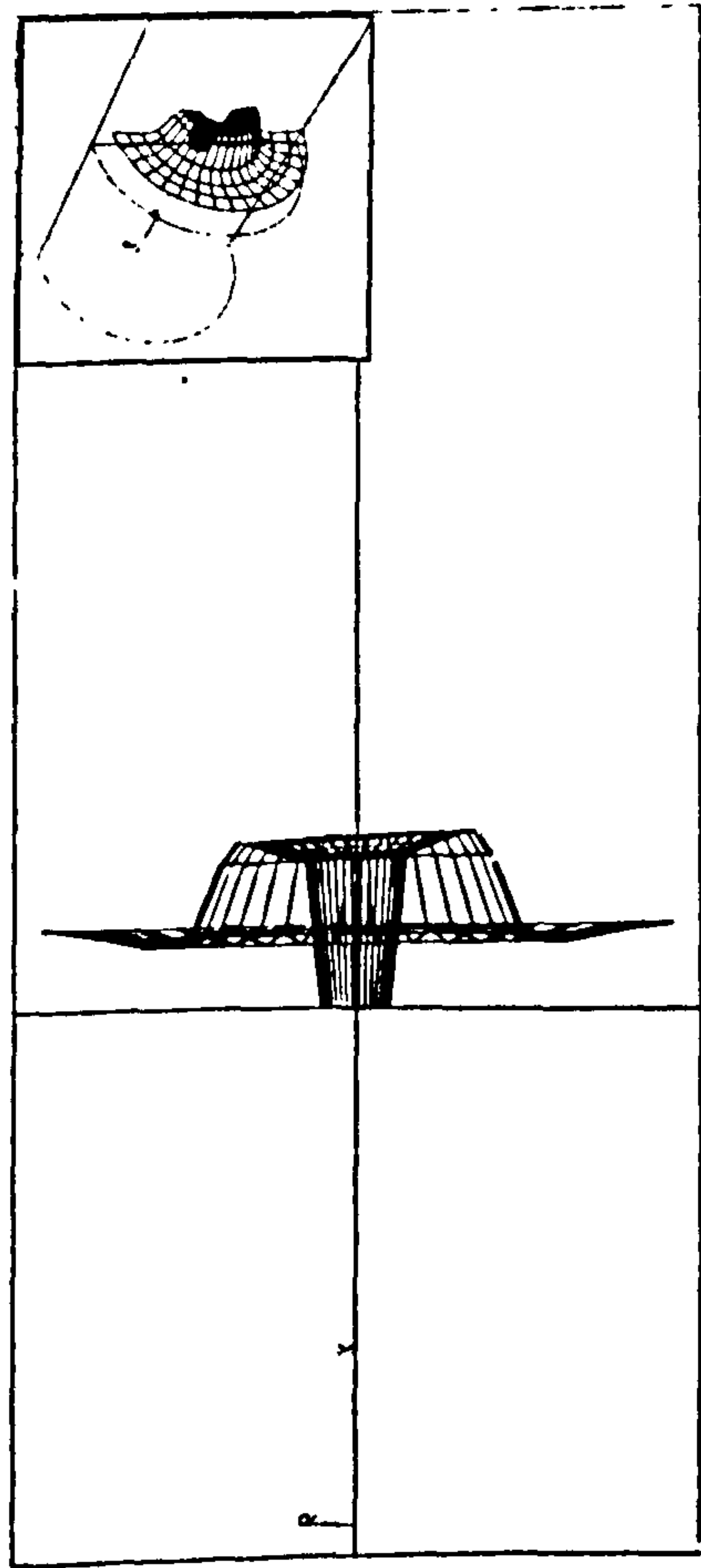
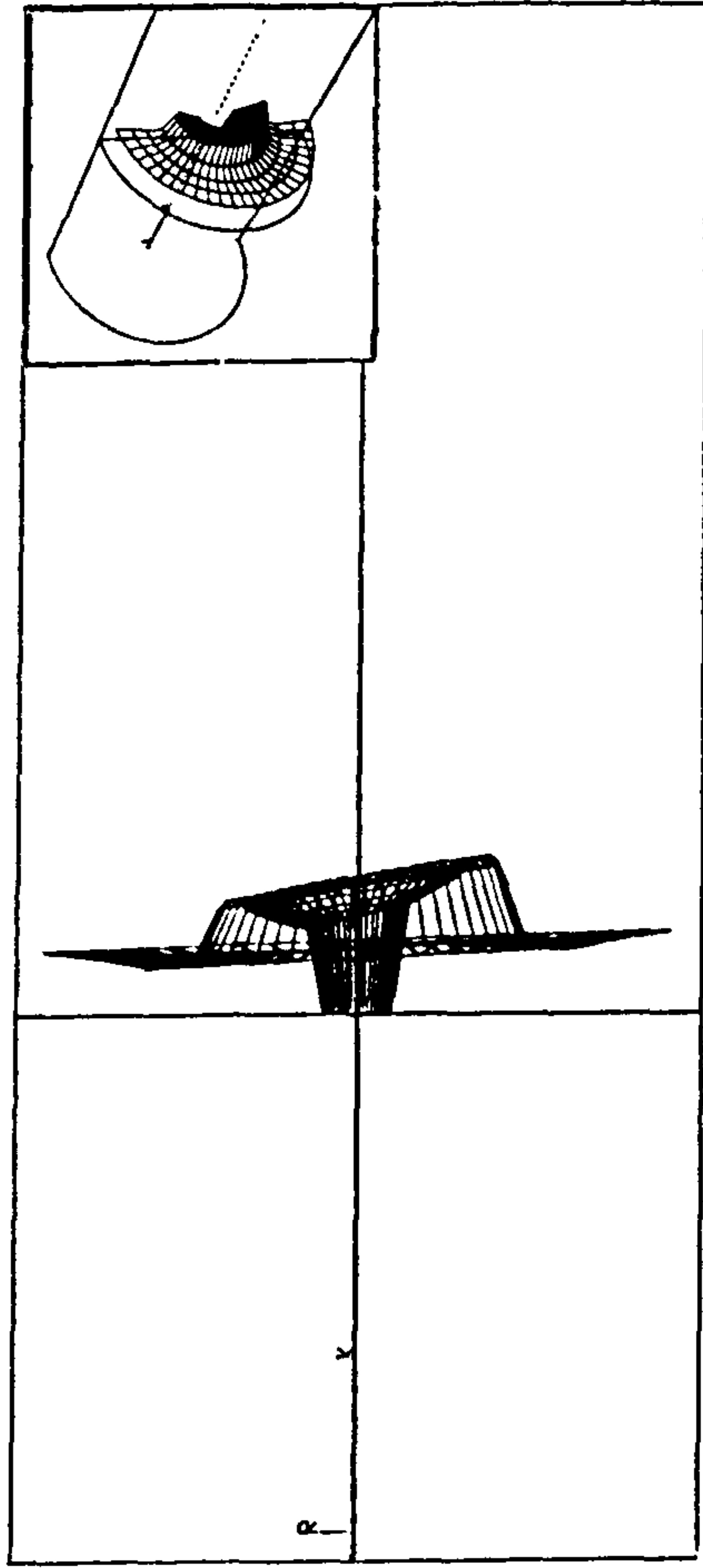


Fig. 6.26 Distribution of velocity vectors in the traverse plane of the inlet valve for butterfly valve angles of 30° (a), 50° (b), 65° (c) and 90° (d).

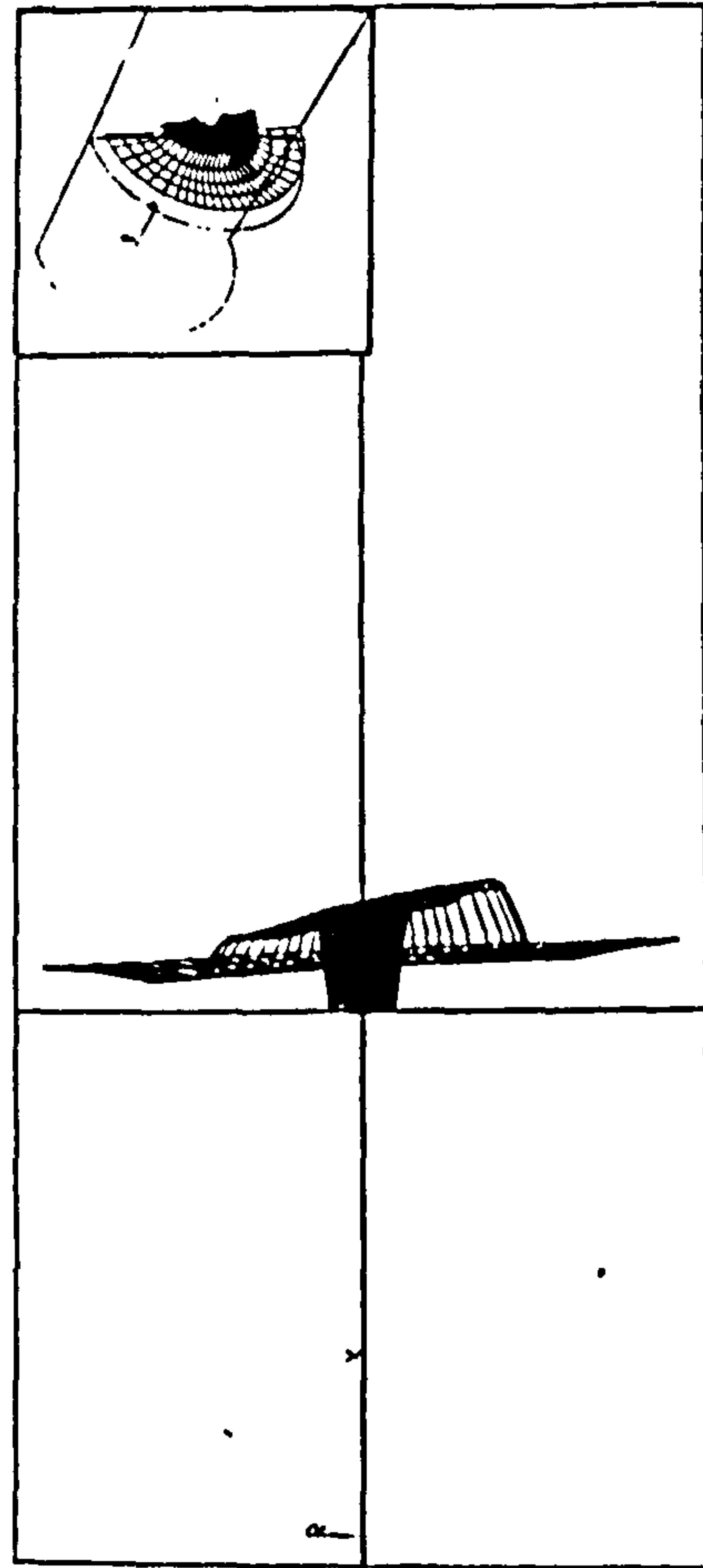
0 0.2 0.4



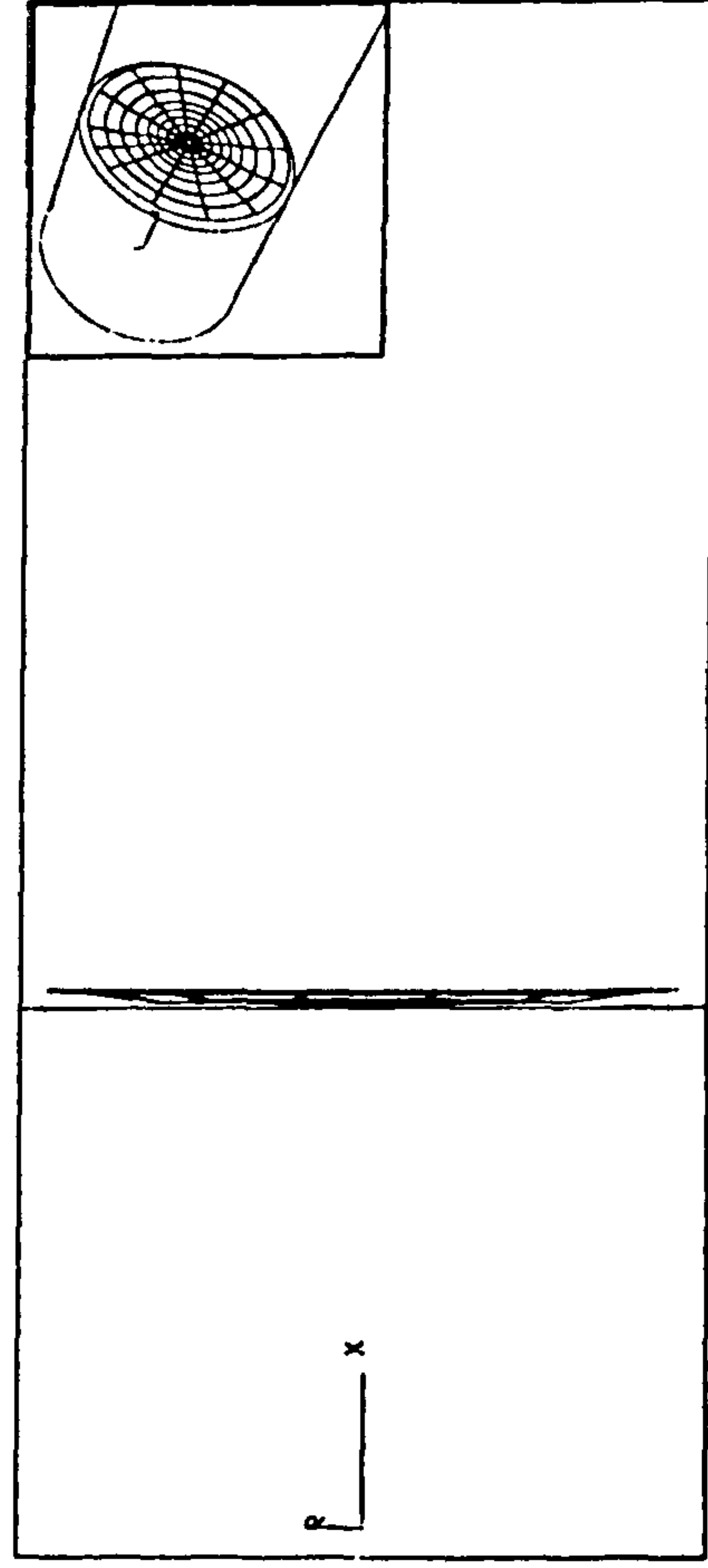
(a)



(b)



(c)



(d)

Fig. 6.27 Distribution of turbulence intensity ($I^2 = \frac{2}{3} K / U_o^2$) at the cylinder head for ramp throttle angles of 30° (a), 50° (b), 65° (c) and 90° (d).

CHAPTER 7COMPUTER MODELS OF IN-CYLINDER UNSTEADY FLOW7.1 Introduction

As was clearly demonstrated in the results of the steady-flow computer simulations, there is considerable potential in these new methods as diagnostic tools. This successful numerical simulation in turn encourages the development of more complete computer models of real engines, with all the attendant complexities. It is, however, important to inject a note of caution into this process by reminding ourselves that the predictions of such methods are subject to particular types of errors, many of which have not yet been carefully quantified. The errors in question result, as outlined previously, from approximations inherent both in the numerical techniques and the mathematical models of the physical processes which the methods embody. Additional uncertainties may be introduced through the boundary conditions appropriate to the particular engine being simulated. It is also important to note that it is not usually possible to make a priori estimates of these effects.

It is equally clear that until some definite statements can be made concerning the accuracy of the predictions of the computer methods, they will continue to be viewed with some scepticism by the engine community.

In this chapter the PHOENICS code has accordingly been applied to a programme of unsteady-flow assessment, involving comparisons with experimental in-cylinder measurements available in the literature. The focus of attention in this chapter is on simulations involving air motion in the absence of combustion and in a non-compressing, relatively simple, axisymmetric model engine; the reason for this being that, from the practical point of view, such systems are far more amenable to accurate in-cylinder measurements than are real engines. Validation cannot satisfactorily be undertaken in the absence of reliability of the experimental data. Additionally, the simplification of axial symmetry substantially reduces the computational costs,

which can be quite high for unsteady-flow cases.

7.2 Source of experimental data

The experimental data which is used for comparison and validation of the computational model was taken from the turbulent flow measurements in a motored piston-cylinder assembly by Morse et.al (85).

In this study Laser-Doppler anemometry was used to quantify the mean velocity and turbulence characteristics of the isothermal incompressible flow within the piston-cylinder arrangement, motored without compression and with idealized inlet geometries corresponding to a pipe and to an annular port located in the center of the cylinder head (Fig. 7.1). This consisted of a mock valve inserted in an oversized seat, so that with the valve face flush with the cylinder head an annular opening remained for the entry and exit of air.

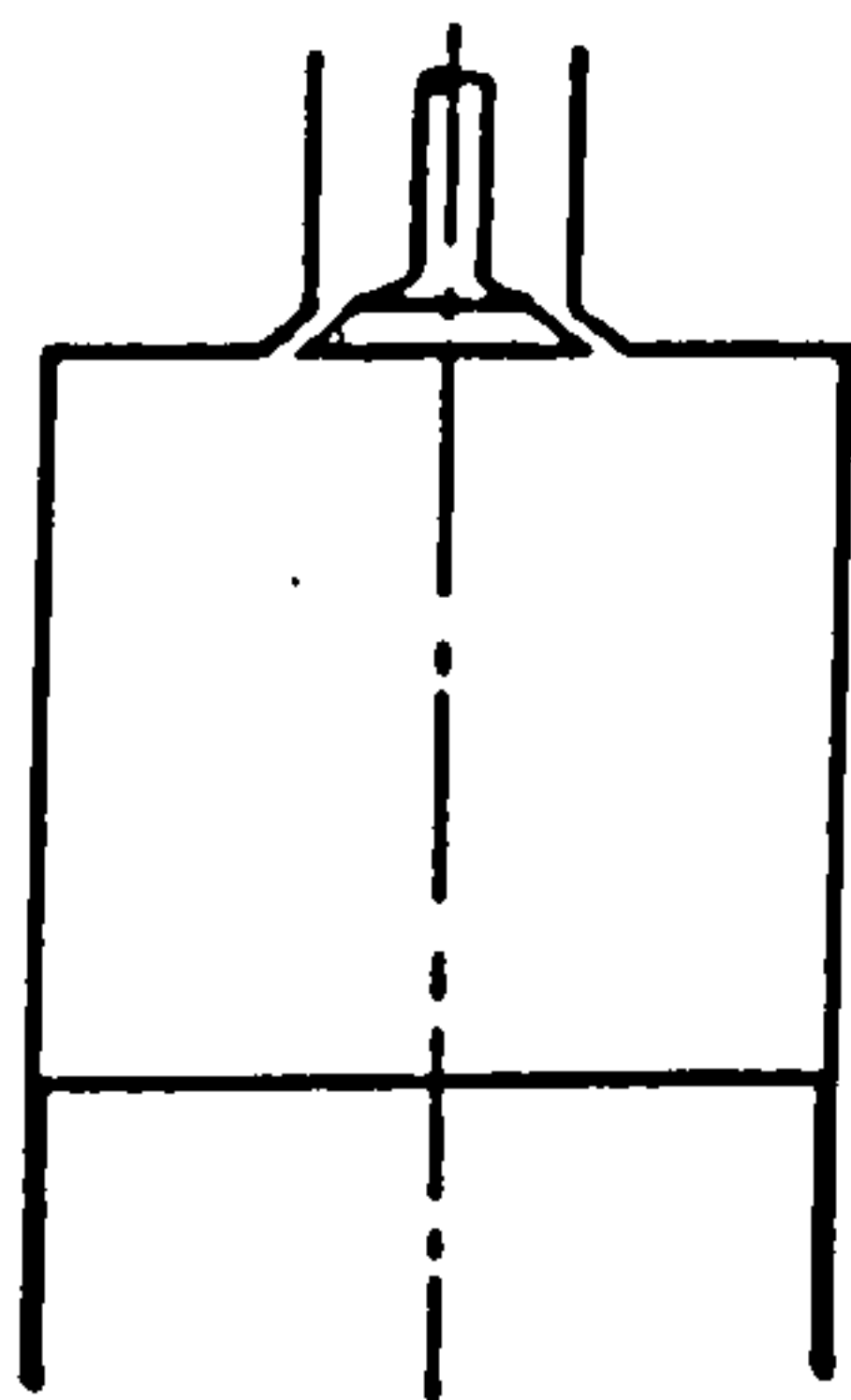


Fig. 7.1 Non-compressing chamber with an annular opening.

7.2.1 Experimental model engine configuration

The model engine had the following dimensions:

- Internal diameter of the cylinder = 75 mm.
- Engine stroke length = 60 mm.
- Ratio of sweep to clearance volume = 2.0
- Annular port angle of entry or valve seat angle = 30° .
- Inner diameter of the port = 33.65 mm.
- Outer diameter of the port = 41.65 mm.

The rotational speed of the model engine was 200 rpm, which was sufficiently high to ensure turbulent flow within the cylinder.

In the absence of any detailed information about the inflow structure into the cylinder three alternative valve geometries were explored in order to observe the effects of inlet port geometry and boundary conditions on subsequent in-cylinder flow structure.

7.3 Two-dimensional unsteady simulations

7.3.1 Use of moving-grids for simulation of engine motion

The EARTH programme of PHOENICS contains a number of moving-grid options which can be used in unsteady flow calculations. In these the grid, or a portion of it is either expanded or contracted in the z-direction, the axial direction of the flow. The expanding/contracting feature is typically introduced through a coordinate transformation in which the axial coordinate Z is replaced by a non-dimensional coordinate, containing information about the instantaneous piston displacement, say $\Omega \equiv Z/Z_p$, Z_p being the instantaneous piston displacement. The cylindrical-polar form of the general governing differential equation (i.e. from the previous chapter) then becomes

$$\begin{aligned} \frac{1}{Z_p} \frac{\partial}{\partial t} (\rho Z_p \phi) + \frac{1}{Z_p} \frac{\partial}{\partial \Omega} (\rho \tilde{u} \phi) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho v \phi) \\ - \frac{1}{Z_p} \frac{\partial}{\partial \Omega} \left(\frac{\Gamma_\phi}{Z_p} \frac{\partial \phi}{\partial \Omega} \right) - \frac{1}{r} \frac{\partial}{\partial r} \left(r \Gamma_\phi \frac{\partial \phi}{\partial r} \right) + S_\phi = 0 \end{aligned} \quad (7.1)$$

where $\tilde{u} = u - \Omega u_p$ is the local relative velocity between the fluid and the coordinate plane, u_p being the instantaneous piston velocity.

In PHOENICS the expansion or contraction is uniform over each constant Z slab during a time step, and the grid is treated as having three regions:

Region 1, from $Z = 0$ to Z_{w1} , is fixed and non-stretching.

Region 2, from Z_{w1} to Z_{w2} , stretches or expands and contracts in a manner to be defined.

Region 3, from Z_{w2} to the end of the domain, which moves by

the same amount as the end of the second Region but does not expand or contract.

In the context of the present model engines, Region 1 represents the inlet port and the clearance volume in the cylinder head, Region 2 is the space between the piston crown and the end of the clearance volume (i.e. swept volume), and there is no need for Region 3, since there is no cavity or bowl in the piston.

In the moving grid option used in the present studies Region 2 expands and contracts according to the formula appropriate to a piston driven by a connecting-rod and crank shaft. The formula employed in PHOENICS (Ref. 65) is

$$(Z_{w2} - Z_{w1}) = (Z_{w2} - Z_{w1})_{t=0} - A (1 - \cos Bt) + C \left[1 - \sqrt{1 - (A \sin Bt / C)^2} \right] \quad (7.2)$$

where A is the crank radius, B is the crank shaft angular speed in radians per second, and C is the connecting-rod length.

7.3.2 Geometric specifications and grid arrangements

The three test cases for which predictions have been obtained with PHOENICS are presented in this section, namely:

Case 1: Non-compressing engine with annular orifice inlet, seat angle of 30°, geometrically similar to the experimental engine.

Case 2: Non-compressing engine with annular orifice inlet, seat angle of 45°. The inner and outer port diameters are identical to those of the experimental engine.

Case 3: Non-compressing engine with annular orifice inlet, seat angle of 30°, with 1 mm thick adapted valve seat, identical to case 1 with the exception of leading edge profile.

These cases were of particular interest since, in addition to experimental data (Morse (85)) there were also computational data (Gosman (86)) available for validation and comparison with PHOENICS predictions.

For all three cases the expanding and contracting portion of the grid was set to span the last 12-grid intervals in the z-direction. The moving and non-moving portions of the grid were arranged

to simulate the conditions encountered in the test engine, in particular the ratio of the sweep to clearance volume was set at a value of two. The geometric information concerning the annular ports were represented in the stationary part of the grid by partial or complete blockage of relevant cells and hence their effective exclusion from the calculation domain. These are illustrated in Figs. 7.2 a to c.

In each case the crank radius, A , was set to 30 mm (i.e. 60 mm stroke length, spanning the last 12-grid intervals), and the connecting-rod length, C , was set to 150 mm.

7.3.3 Incorporation of boundary conditions

The boundaries to be considered were of two kinds, namely, the confining walls and the plane of inlet at $Z = 0$.

Wall boundary conditions were identical to the ones used in the steady flow cases (cf. Chapter 6) and were applied to the walls of the inlet pipe, cylinder head, valve face, cylinder and piston crown. Wall boundary conditions could not be specified on the inner and outer annular port walls, since they were represented by partially blocked cells.

At the inlet plane ($Z = 0$) the relative pressures in all cells were set close to zero. In the absence of details of the inflow (i.e. not provided by the experimental data (Ref. 85)), the mass inflow, velocity and turbulence values were simply based on piston displacement calculations, so for example the indraw jet has a velocity slightly less than ten times that of the piston.

Given details of chamber configuration (section 7.2.1), piston motion (section 7.3.1) and boundary conditions, PHOENICS solves the difference equations on a prescribed grid by a time-marching process in which solutions are obtained at small intervals of time or, equivalently, crank angle, until the desired period has been covered. For each new time interval the method updates the fields of variables from the preceding interval.

The engine speed for all three cases was set at 200 rpm (6.667 rad/sec). The starting time of the calculations was set at TDC (engine Top Dead Center). The initial field of all the variables at TDC was set at default values. The finishing time of the calculations was set to correspond with 144° ATDC (After Top Dead Center). This time

interval, which corresponds to a crank advance of 144° , was subdivided into 34 steps distributed as follows: 12 steps each of 3° , 1 step of 4° , 10 steps of 5° , 1 step of 4° and 10 steps of 5° .

7.3.4 Convergence and grid dependence

In each of the three cases, in which 34 time steps were taken, around 30 domain sweeps at each time step were found sufficient to bring the total continuity errors to very small values, less than 0.1% of the total mass inflow at each time step. When the errors were small, and the monitor values unchanging, it was concluded that the solution was satisfactorily converged.

No formal grid-refinement studies were performed, since the process would have been both costly and time consuming. Case 3 -with its finer grid distribution around the annular port (see Fig. 7.2 c)- was, however, run without the leading edge geometry (geometrically identical to Case 1), and the results were found to be very similar if not identical to those calculated for Case 1.

7.4 Discussion of results

The behaviour of the measured flow is illustrated by the stream line patterns in the sequence of plots shown in Fig. 7.3, taken from Ref. 85. It is immediately apparent, especially in the earlier phases of intake, that the entering annular jet assumes a steeper angle than that of the valve seat, at least in the region near the opening accessible to measurements. Separation occurs both at the inner and outer edges of the annulus and results in two vortices of unequal size and strength being formed at 36° ATDC, with the larger of the pair located in the lee of the valve and the smaller one positioned near the cylinder head corner. The jet, meanwhile, flows between these two vortices and impinges almost normally on to the piston.

Later, at 90° ATDC, the central eddy has enlarged and strengthened at the expense of the corner one, and the jet now follows a shallower angle, closer to valve seat angle but still larger. At the same time, the adverse pressure gradient brought about by impingement on the piston is apparently sufficient to cause the cylinder-wall boundary

layer to separate and form a third vortex near the piston. This behaviour persists at 144° ATDC where the main vortex has further increased in strength and the one near the piston has also grown larger.

The behaviour of the computer predicted flow for Case 1, corresponding to a valve seat angle of 30°, is illustrated by the velocity vector plot sequences in Fig. 7.4. (Please take notice of the non-expanding nature of the vector plots in Figs. 7.4 to 7.6. The axial scaling however, provides the necessary information about the actual dimensions at relevant crank angles). They also show the entering jet assuming a shallower angle, very close to that of the valve seat, resulting in a smaller and weaker eddy near the cylinder head corner. This is particularly significant in the early phases of intake, for example at 36° ATDC. However, there appears to be no indication of flow separation or recirculation at the cylinder wall at 90° or 144° ATDC. Most of the dissimilarities between prediction and experiment appear to have been caused as a result of the shallower entry of the annular jet. This is further demonstrated in velocity vector plots of computer prediction for Case 2 (i.e. valve seat angle of 45°), shown in Fig. 7.5. The steeper entering jet angle results in a larger corner vortex near the cylinder head at 36° ATDC. This vortex appears, however, to be weaker than its experimental counterpart. The separation of the cylinder-wall boundary layer and formation of third vortex near the piston, which was missing in Case 1, is also clearly indicated for crank angle 90° and 144° ATDC.

In test Case 3, with the valve seat angle at 30° and a 1 mm chamfer on the leading edge, the diversion of the entering jet by the 1 mm leading edge, which is shown in Fig. 7.6, appears to have a similar effect to that of increasing the valve seat angle. At the earlier phase of intake, for crank angle 36° ATDC the angle of entering annular jet appears to be even steeper than that of its counterpart for Case 2 (i.e. valve seat angle of 45°). This is accompanied by a larger corner vortex near the cylinder head and a stronger jet flow between the center and corner vortices. In the later phases of intake, namely 90° and 144° ATDC, there is a clear indication of cylinder-wall boundary layer separation and recirculation which, if anything, appears to be stronger and cover a larger portion of the cylinder wall than its counterparts in Case 2.

A further diagnostic of the distinctive in-cylinder flow patterns can be seen in the plots of cylinder wall static pressure coefficients (Figs. 7.17 to 7.19) where the reference pressure, P_0 , is the wall static pressure at the cylinder head. The pressure variations along the cylinder wall are clearly very small, $O(10^{-3})$. This reflects the relatively low in-cylinder velocities involved. The detailed flow structure, however, appears to be quite sensitive to these small variations. As can be seen from Fig. 7.19, for a crank angle of 144° ATDC, the region in the vicinity of the dividing stream between the corner and core vortices for valve angles of 45° and 30° with 1 mm edge (i.e. cases 2 and 3) are characterized by an initial positive rise as the flow decelerates approaching the impingement point, and is then followed by a fall in pressure coefficient accompanying the flow acceleration beyond this point. This is then followed by a further rise in pressure coefficient and a relatively stagnant region, indicating further deceleration of the flow leading to cylinder-wall boundary layer separation. On the other hand, the pressure coefficients for a valve angle of 30° (i.e. Case 1) indicate a smaller corner vortex followed by a region of accelerating flow up to about 30 mm downstream of the cylinder head, as evidenced by the steep fall in pressure coefficient. The following region of decelerating flow extends to the piston crown, but apparently does not lead to any flow separation. Similar behaviour can also be observed in Fig. 7.18 at a crank angle of 90° ATDC. At a crank angle of 36° ATDC (Fig. 7.17) some major differences with cases 2 and 3 appear to have been caused by the extent and strength of the corner vortex. Broadly, similar variations in wall static pressure coefficients are observed. Overall the variations in wall static pressure coefficients complement the in-cylinder flow behaviour observed from the velocity vector diagrams, but do additionally demonstrate a high degree of sensitivity to small differences in wall pressure at these low Mach numbers.

The overall impression from the velocity vector plots was that the computer predictions appear to agree relatively well in gross features with the experimental results for cases 2 and 3. It is possible to see, amongst other similarities, the initial formation of twin-vortex structure and emergence of third vortex in the later phases of intake.

For a more accurate assessment of the computer predictions, reference should be made to profile comparisons of the axial velocity and turbulence intensities in Figs. 7.7 to 7.16.

The plots of Figs. 7.7 to 7.9 show comparisons of measured and predicted axial velocities and turbulence intensities for the earlier phase of the intake at a crank angle of 36° ATDC. The predicted axial velocity profiles for Case 2 and Case 3 appear to match the measured values more closely than Case 1. For all three cases the predicted jet appears to give approximately the right trajectory, but to spread and decay too rapidly, especially in the predictions of Case 1. This is accompanied by low shear-generated turbulence.

The plots for 90° ATDC (Figs. 7.10 to 7.13) show a steeper predicted jet trajectory for Case 2 and Case 3 and a much shallower and more widely spread one for Case 1, in comparison with the measurements near the valve (Fig. 7.10). The velocity profiles also give no indication of a reverse flow region near the cylinder wall for Case 1. This seems mainly due to the smaller predicted corner vortex near the cylinder head. Further downstream of the valve, however, the predicted axial velocity profile for Case 1 appears to match the measurements better than those for Case 2 or Case 3 (cf. 30 mm downstream of the valve, Fig. 7.11). The predicted turbulence intensities continue to be much lower than the measurements. This can be partly a result of lower predicted velocity gradients and more rapid spread and decay of entering jet flow. This is especially apparent in the predictions for Case 2 and Case 3.

At 144° ATDC the plots of Figs. 7.13 to 7.16 show better agreement for predicted velocities near the inlet for Case 2 and Case 3, but the predicted jet appears to decay and spread rapidly downstream of the valve. For Case 1 the predicted jet has a shallower angle than the measurements, which in turn gives rise to higher velocities along the cylinder wall. The turbulence intensities continue to be under-predicted.

The continuous under-prediction of shear-generated turbulence intensities is partly due to the lower velocity gradients associated with the rapid decay and spread of the entering annular jet, and partly due to the absence of wall friction or any boundary specification of turbulence quantities in the annular port.

The overall picture which emerges seems, therefore, to be one of strong sensitivity to annular inlet flow conditions and predominantly inlet flow angle. At this point it might be appropriate to refer to similar predictions by Gosman et al (86), in which the importance of inlet boundary conditions were again endorsed. In his predictions, however, the annular jet inlet boundary conditions were specified by fixing the velocity vectors and appropriate turbulence quantities at the inlet plane of the annular jet. His predictions, therefore, relied more heavily on detailed inlet boundary information, which were not available at the time. The present approach, with its reliance on inlet port geometry, appears to be a more promising way forward since, in principle, it can be applied to an arbitrary geometry. The use of a strict cartesian or polar coordinate system and representation of specific geometries by partial blockage of relevant cells seem however to be of limited value. This has been particularly restrictive in the present cases, since the absence of wall friction and shear flow inside the annular port, which in turn has a substantial effect on the entering jet and turbulence quantities, has been totally neglected.

Using body-fitted coordinates (topologically cartesian) to fit the calculation mesh to the geometry of the simulated flow boundaries can overcome many of the shortcomings of the present study, by allowing a more accurate incorporation of, amongst other aspects, wall friction on the enclosing boundaries. The effect such manifestations might have on the annular jet angle at entry and turbulence intensities, however, remains to be investigated.

7.5 Concluding remarks

The turbulence flow field in a non-compressing and axisymmetric model engine have been modelled computationally and compared with measurements by Laser-Doppler anemometry in a motored piston-cylinder assembly. Three alternative inlet configurations have been considered. The following are the main points to emerge from the present assessment exercise:

- The overall structure of in-cylinder flow is apparently strongly sensitive to annular inlet flow conditions, and in particular

to the inlet jet trajectory.

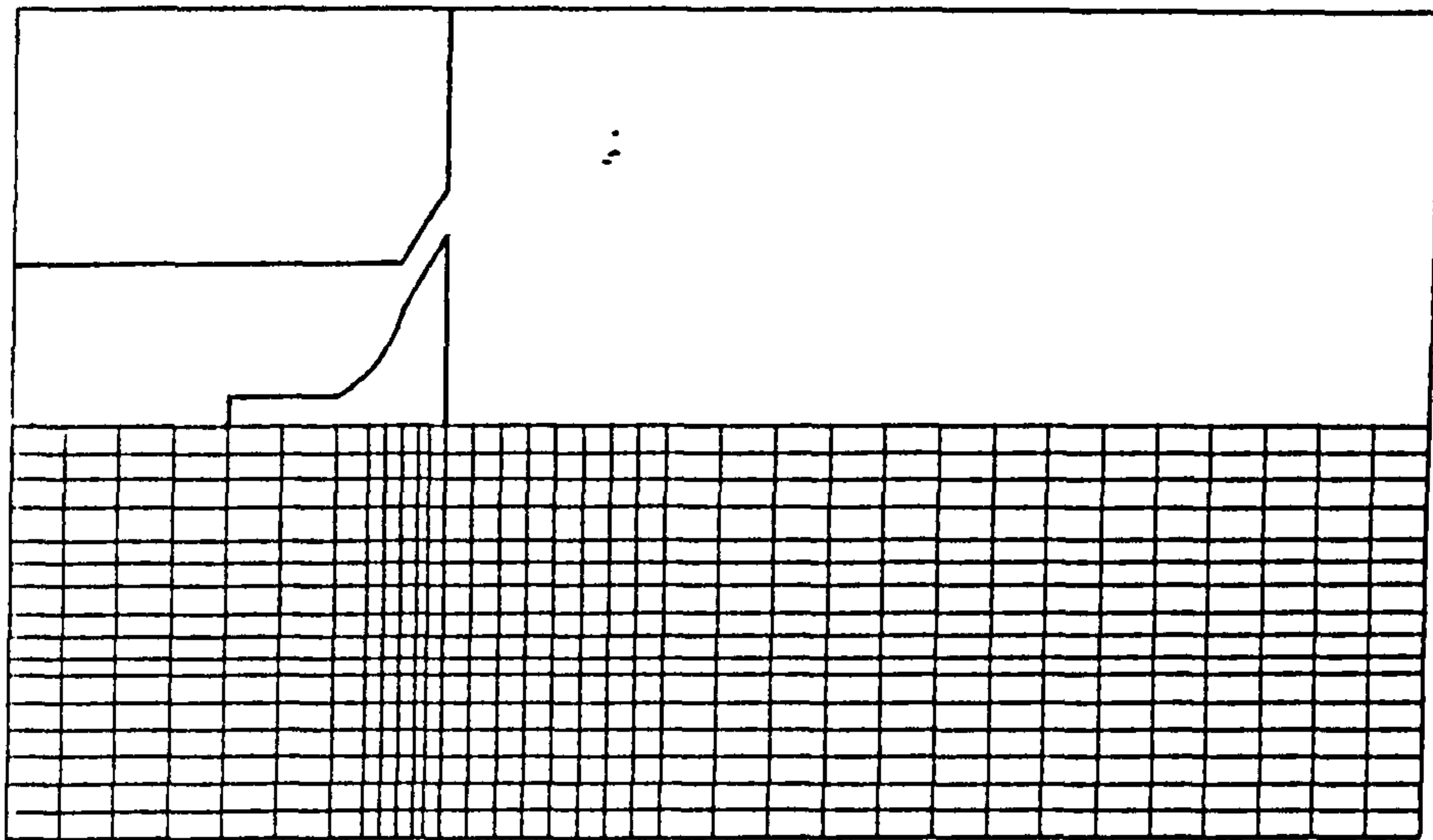
- The steeper annular jet angle entering the cylinder (relative to the valve seat), which was clearly indicated in the measurements, is hardly noticeable in the computer predictions. This might be partly due to the absence of wall friction on the annular port walls in computer simulated models.

- The predictions for steeper valve seat angle or flow diverting valve with a non-zero leading edge seat thickness appear to match the measured flow fields better, with closer predicted inlet jet trajectories in particular.

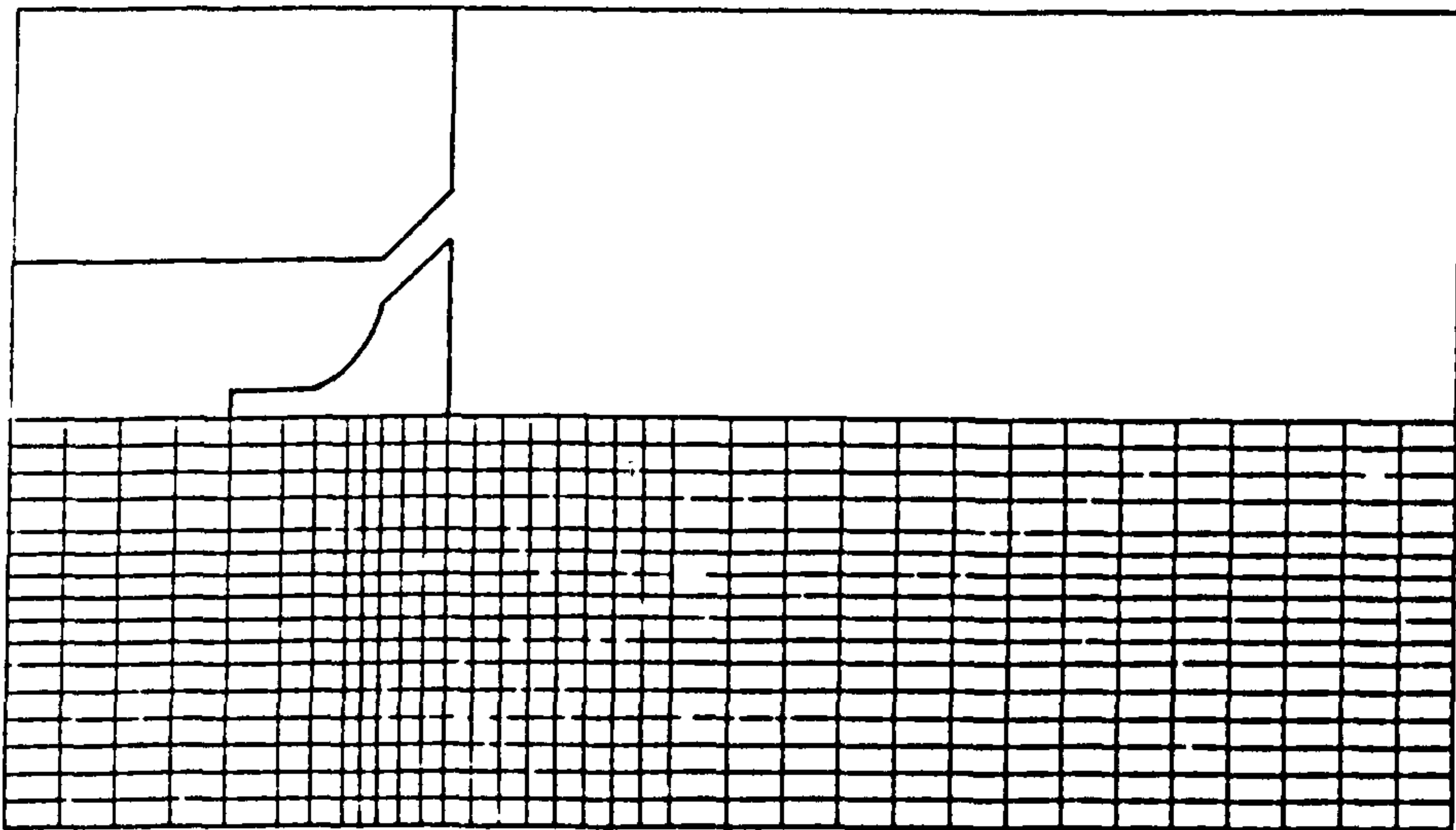
- All the predictions show a more rapid decay and spread of the entering jet in comparison with their experimental counterparts. This, in turn, results in lower predicted velocity gradients.

- The shear-generated turbulence intensities are persistently under-predicted and are not very satisfactory. This appears partly due to the lower velocity gradients and partly the absence of wall friction or satisfactory turbulence quantities by way of boundary condition inside the annular port.

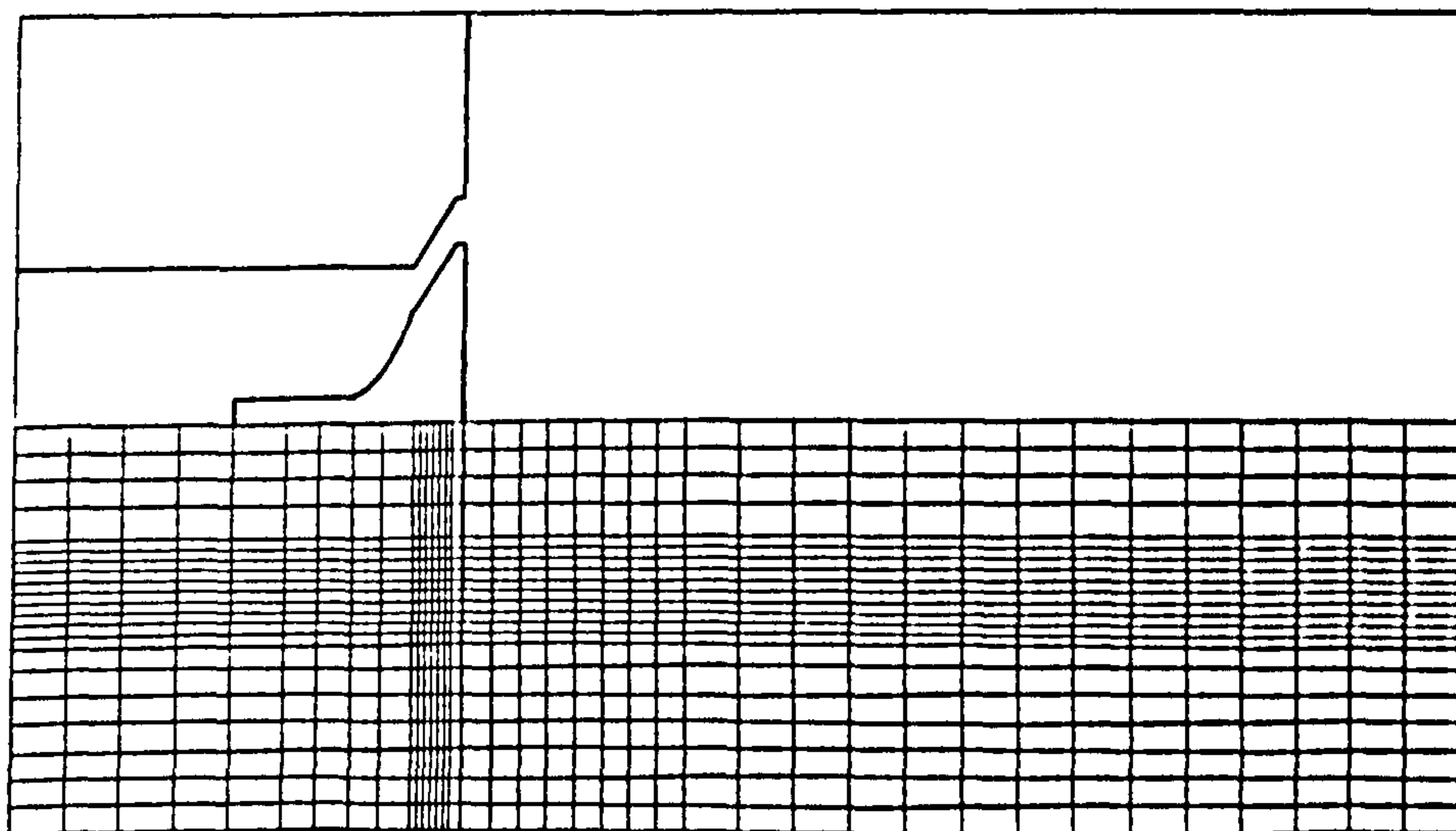
- Incorporation of appropriate boundary conditions inside the inlet port is made possible by the use of a body-fitted coordinate system, now available in PHOENICS but not when the study was initiated. Until such times as further studies are undertaken, the net consequences of other defects on overall air flow prediction can not be fully quantified. In the meantime, however, PHOENICS does already provide a most useful insight to the engine analyst; one which has been substantially validated by comparison with experiment.



Case 1



Case 2



Case 3

Fig. 7.2 Computer simulated port geometries and grid distributions for Case 1 (i.e. 30° seat angle), Case 2 (i.e. 45° seat angle) and Case 3 (i.e. 30° seat angle with 1 mm leading edge).

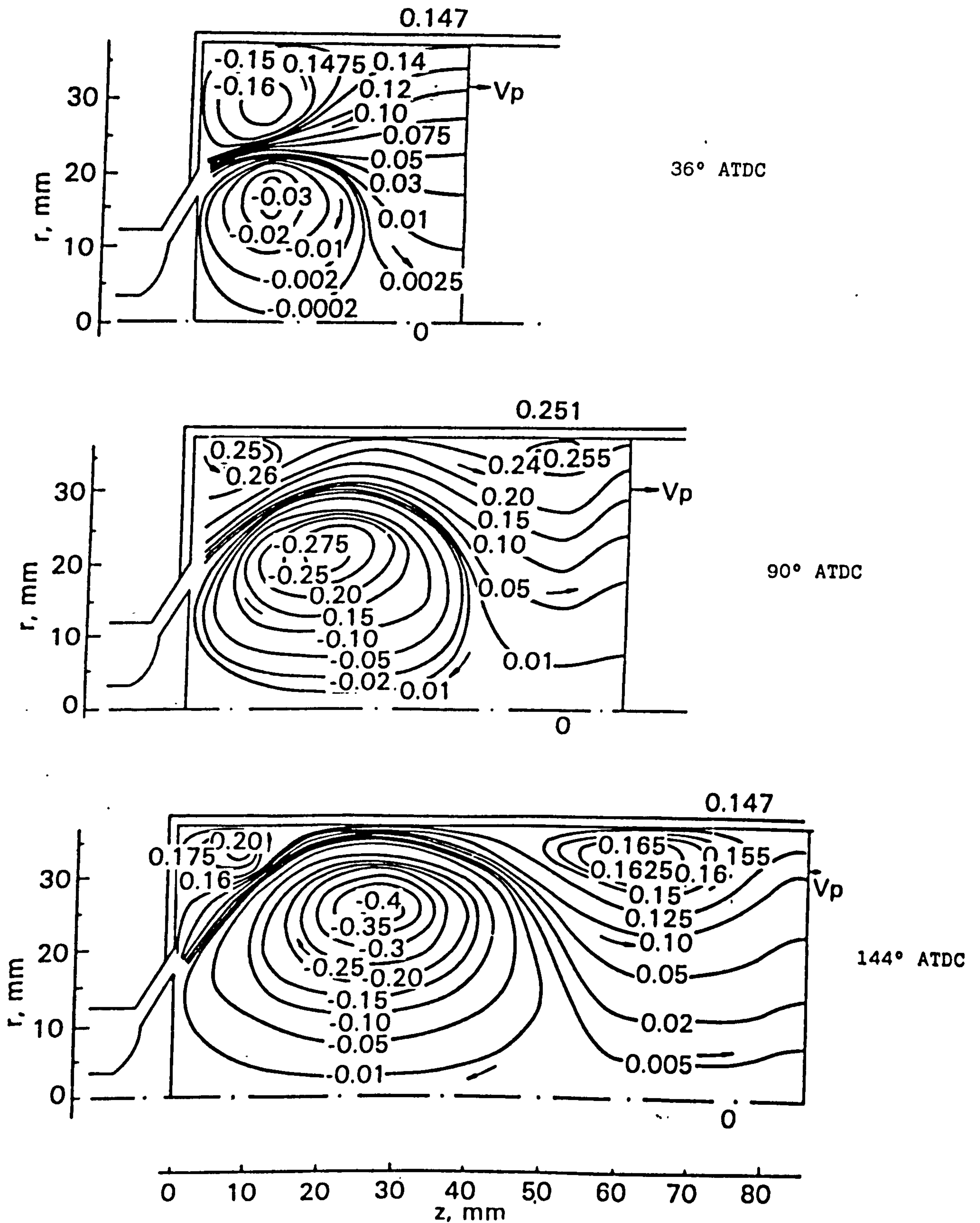


Fig. 7.3 Measured velocity fields for non-compressing model engine with orifice inlet (Ref. 85).

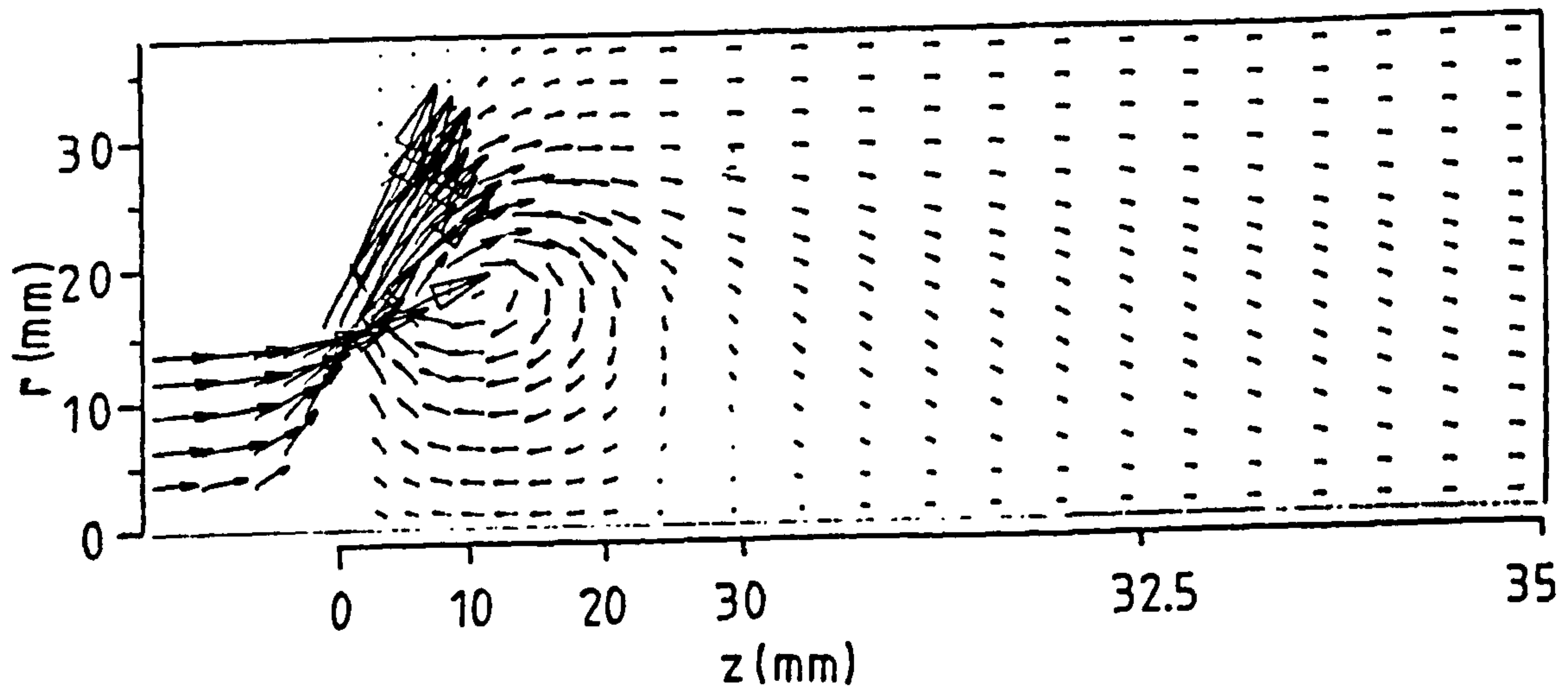
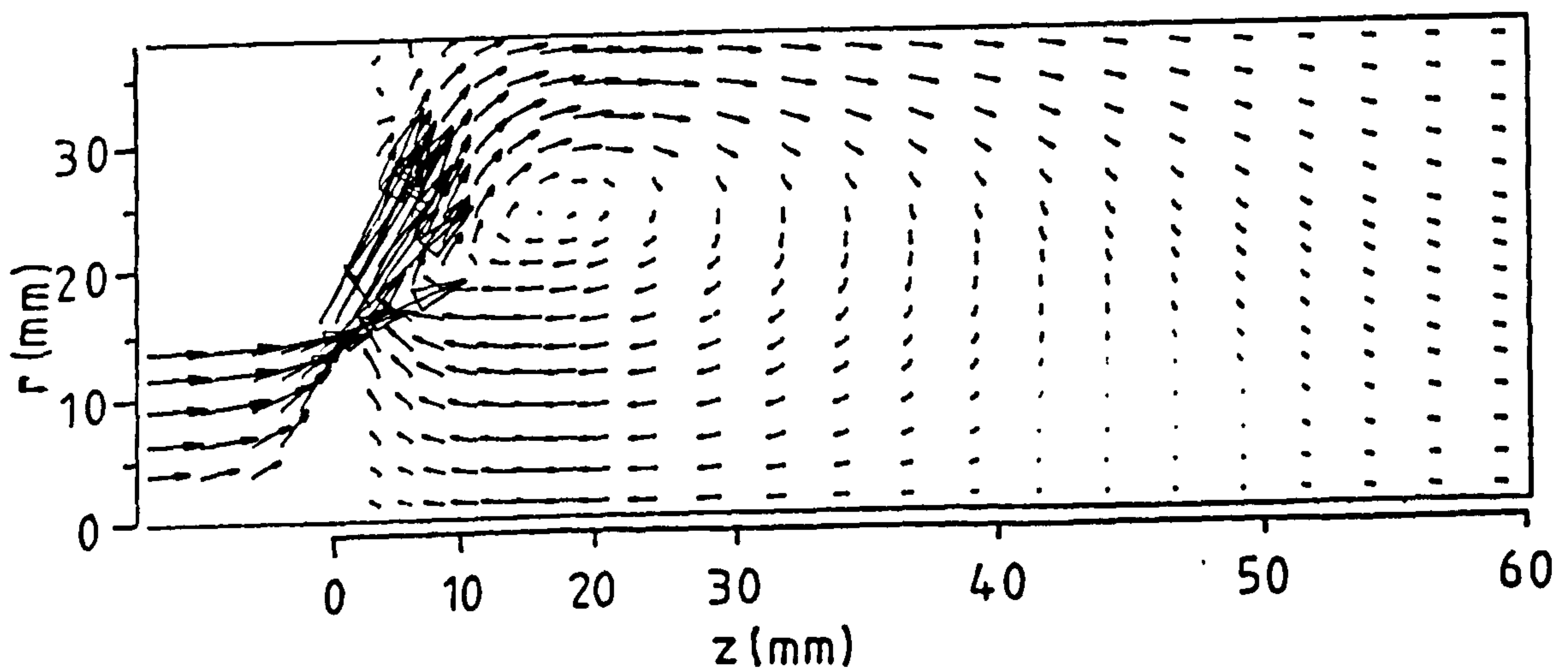
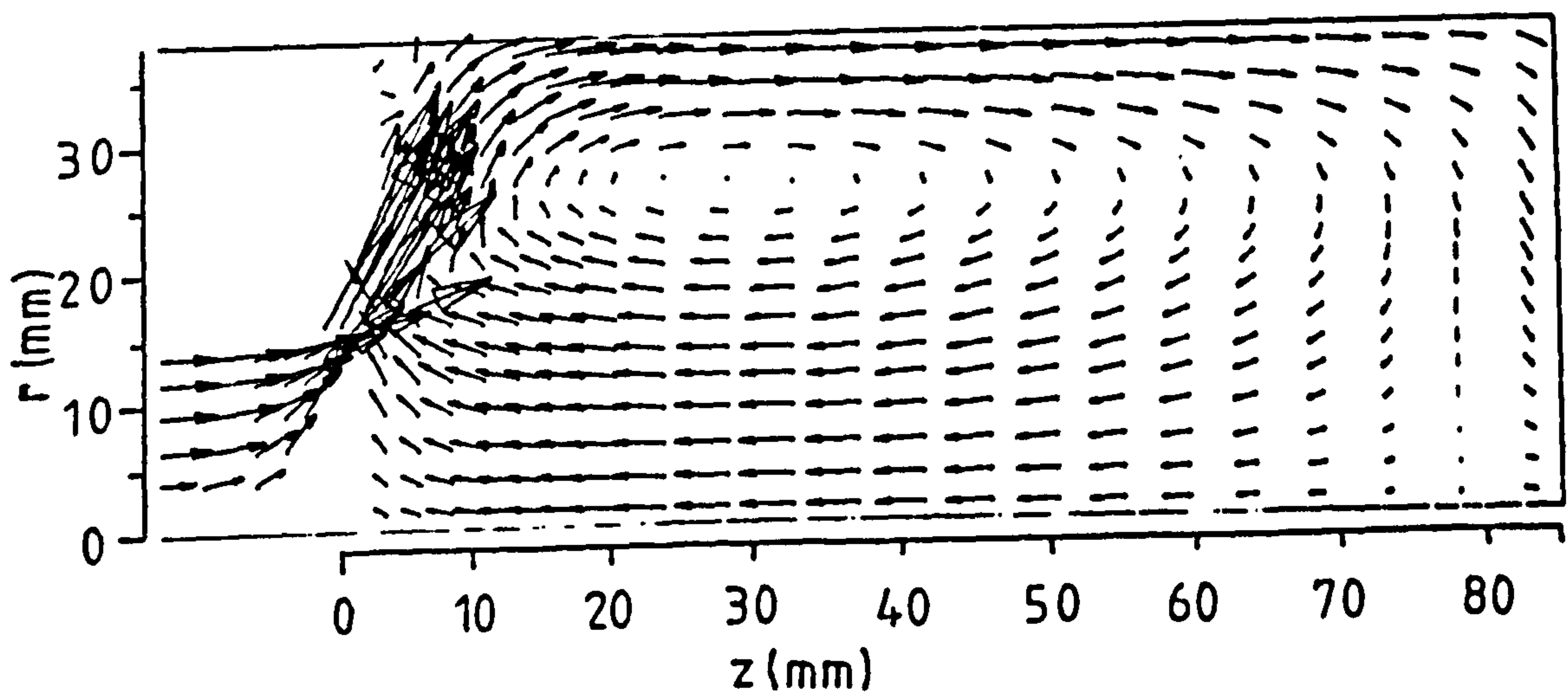


Fig. 7.4 Predicted velocity fields for Case 1 with inlet valve angle specified at 30°.

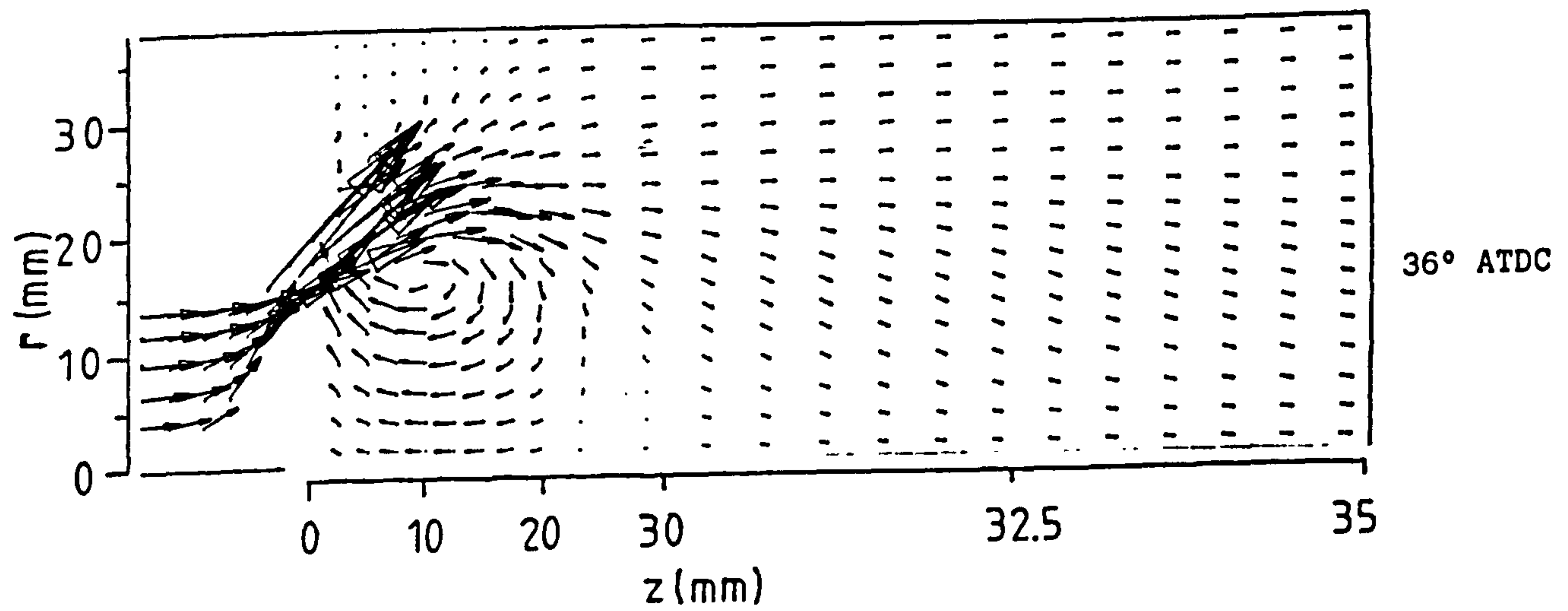
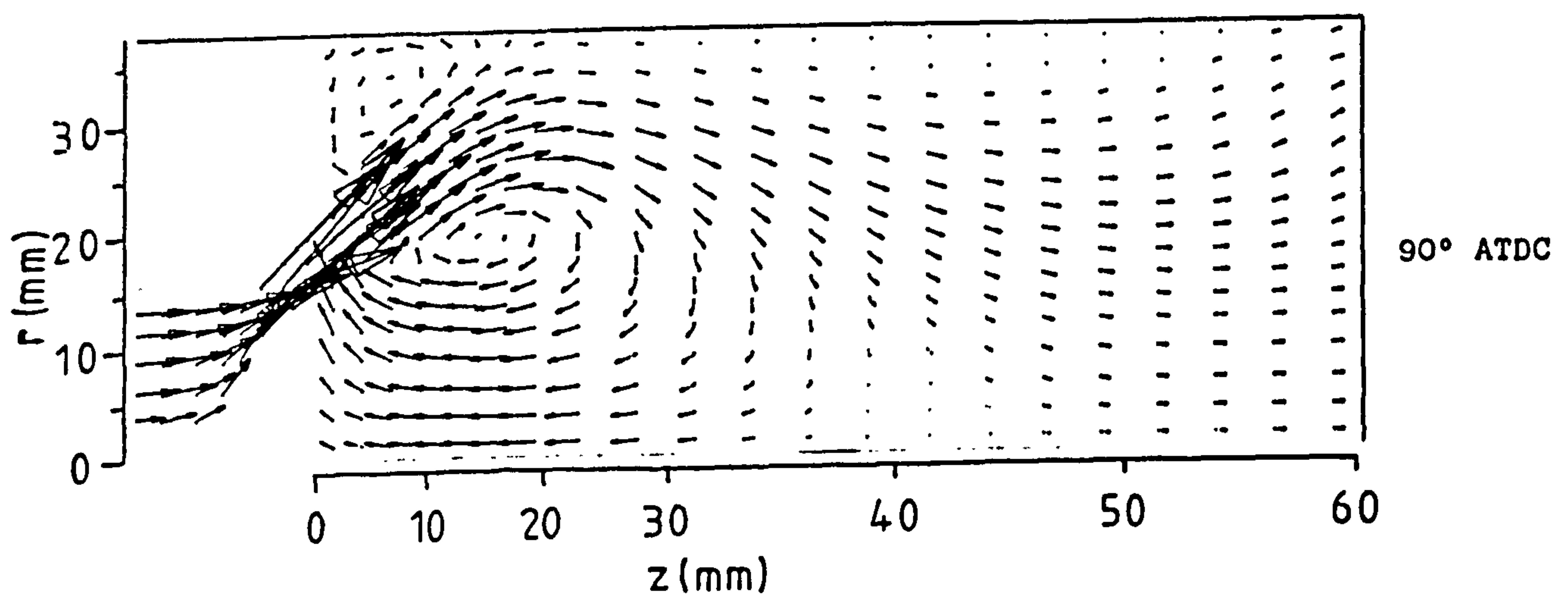
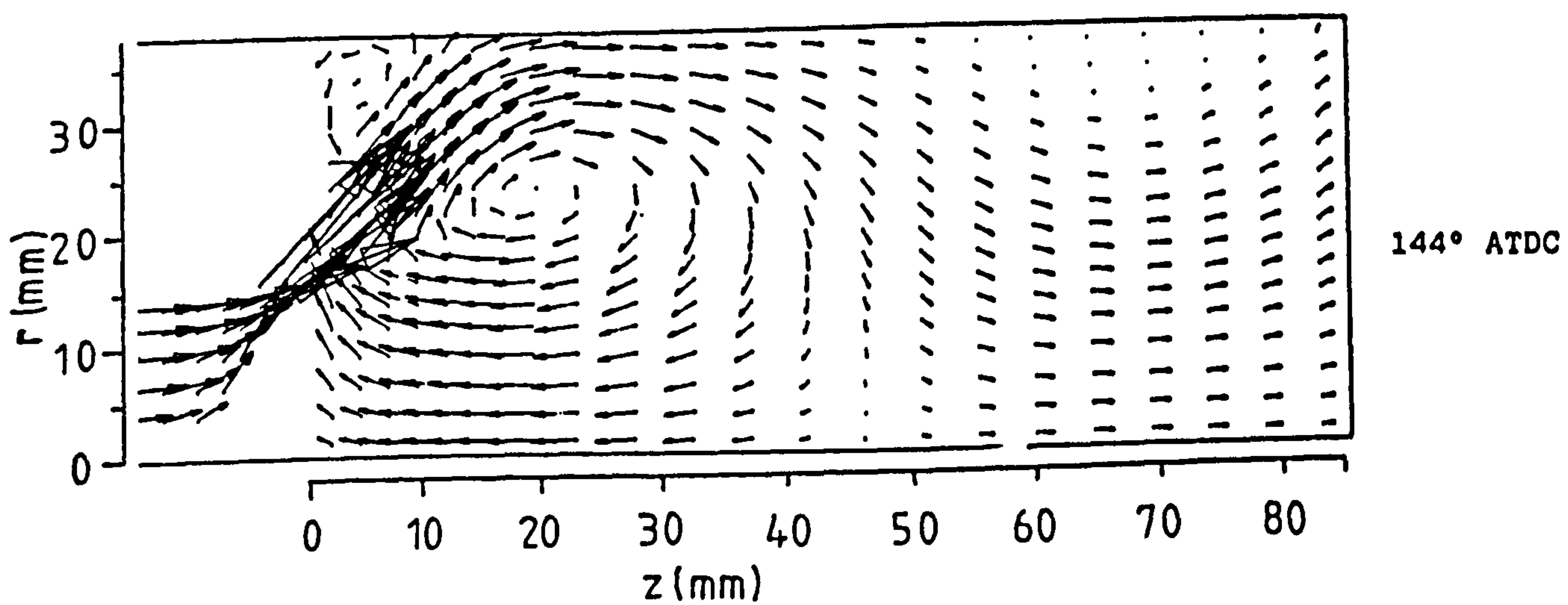


Fig. 7.5 Predicted velocity fields for Case 2 with inlet valve angle specified at 45° .

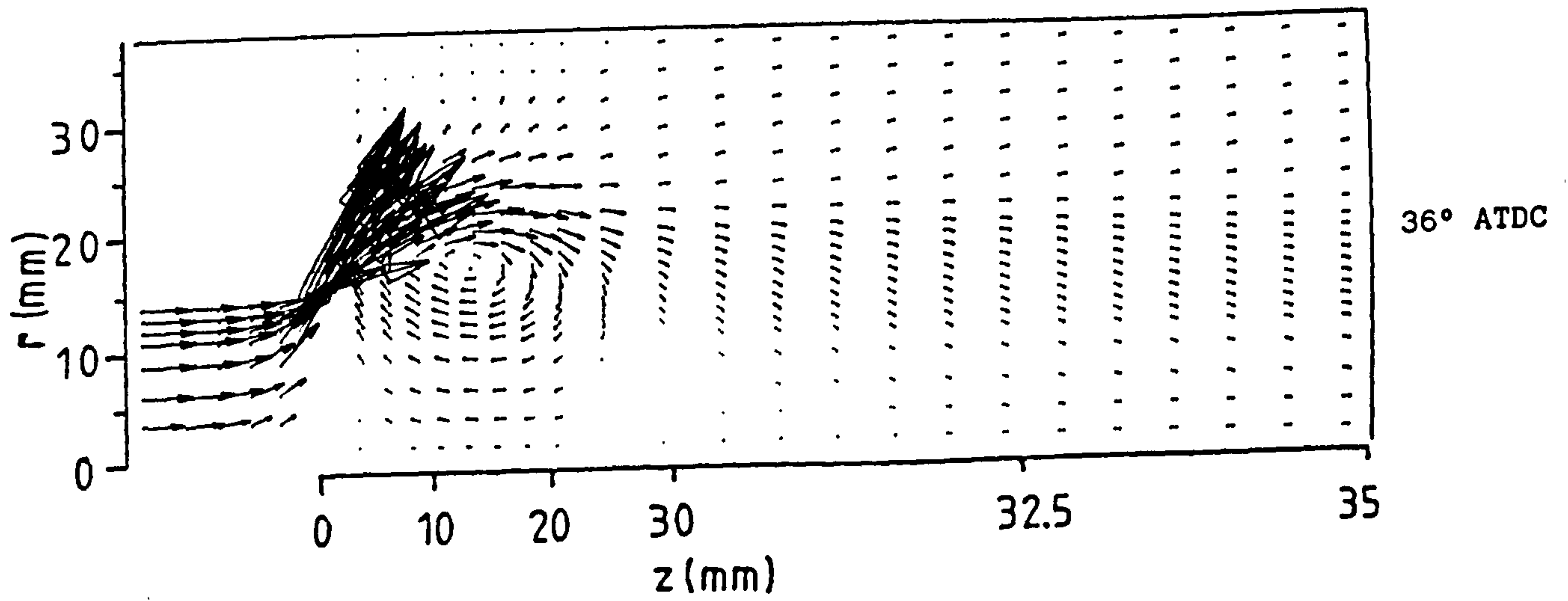
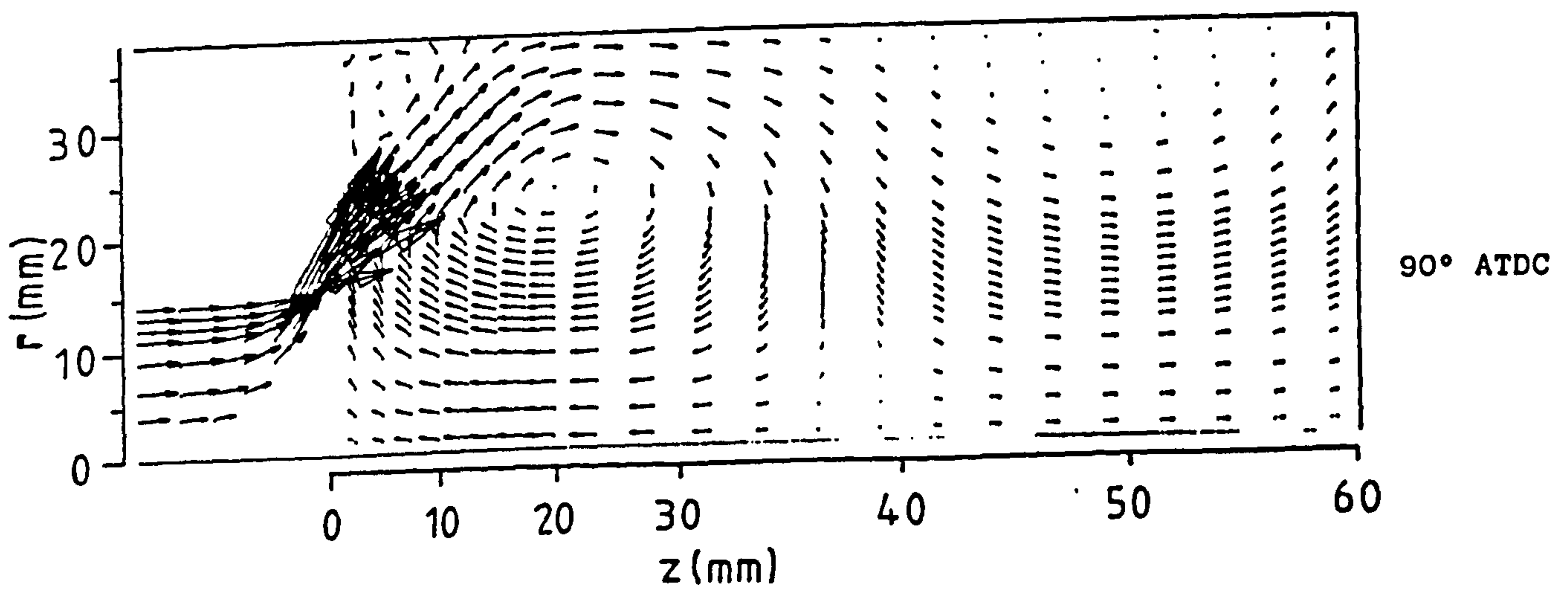
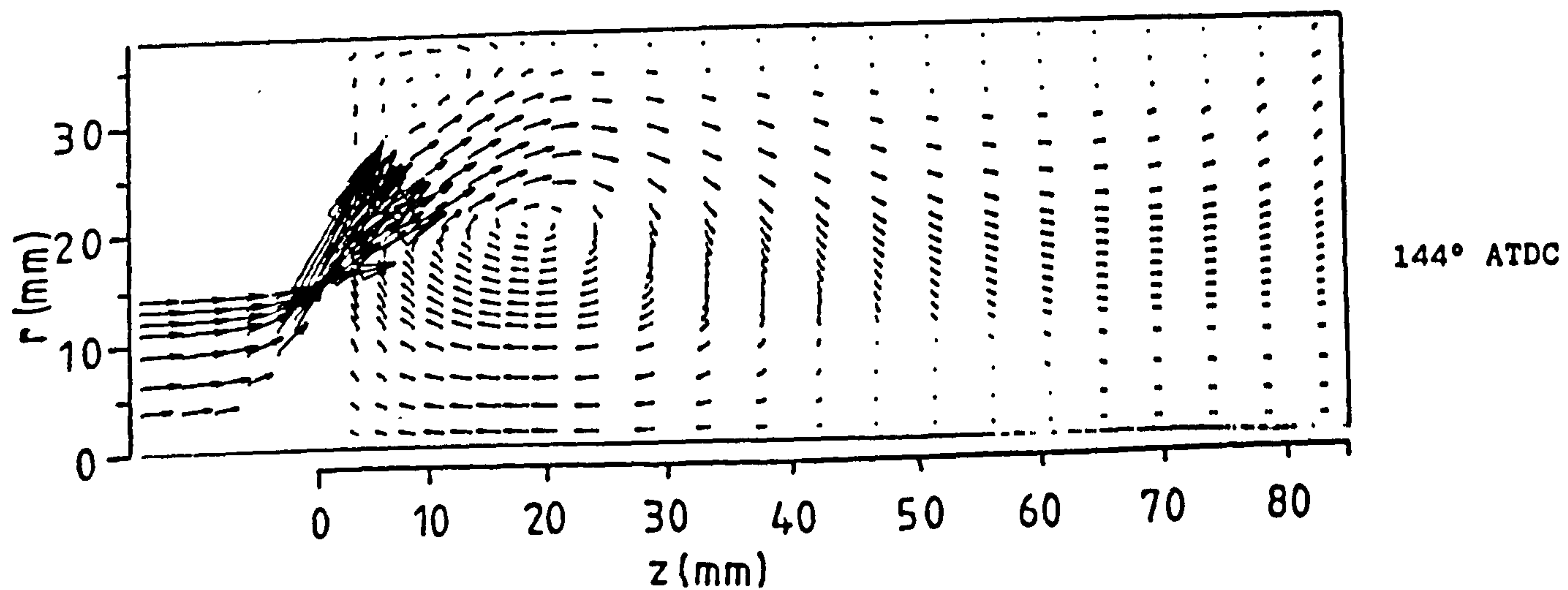


Fig. 7.6 Predicted velocity fields for Case 3 with inlet valve angle specified at 30° and a 1 mm leading valve seat.

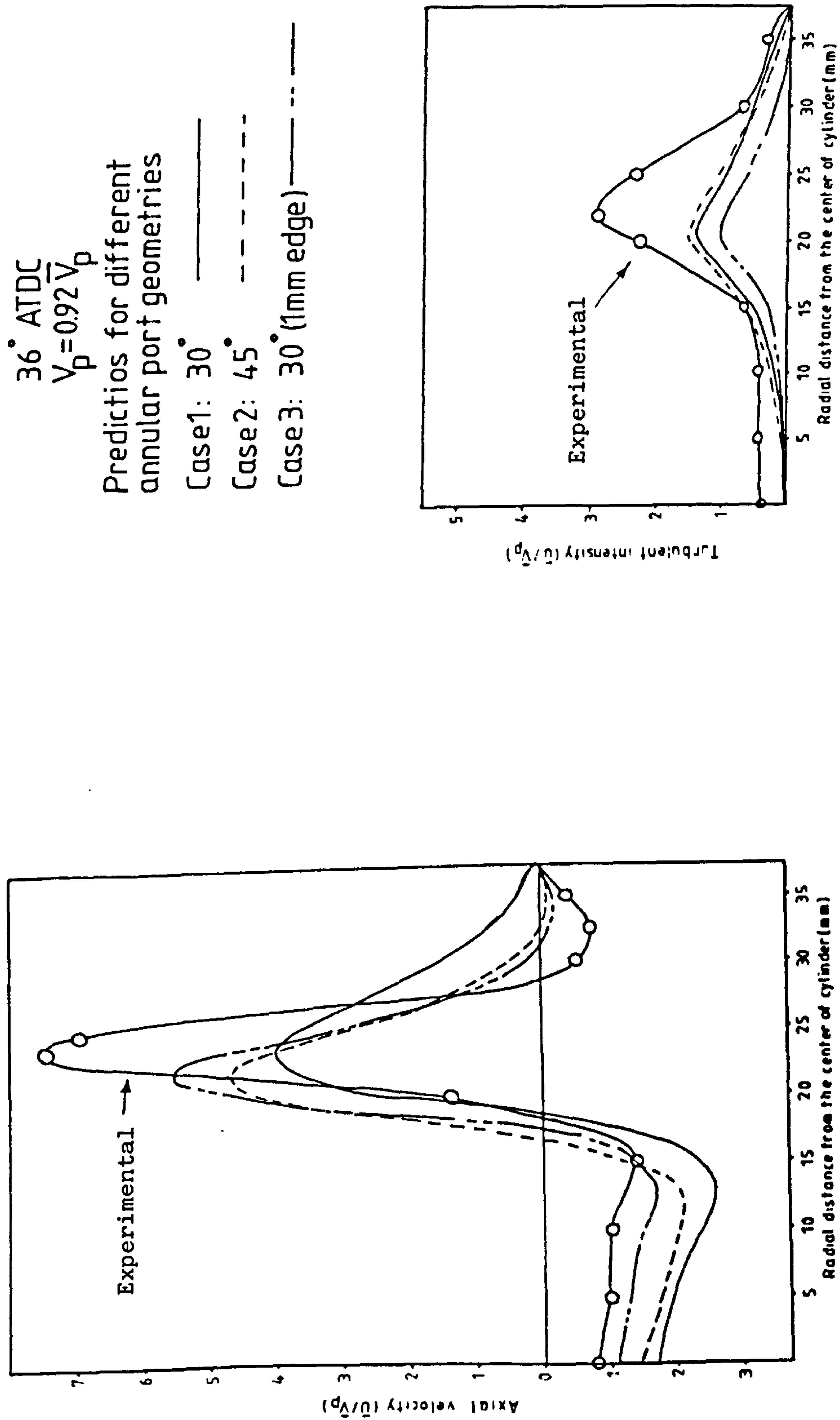


Fig. 7.7 Comparisons between measured and predicted axial velocity and turbulence intensity profiles at 10 mm downstream of the annular valve, for crank angle 36° ATDC.

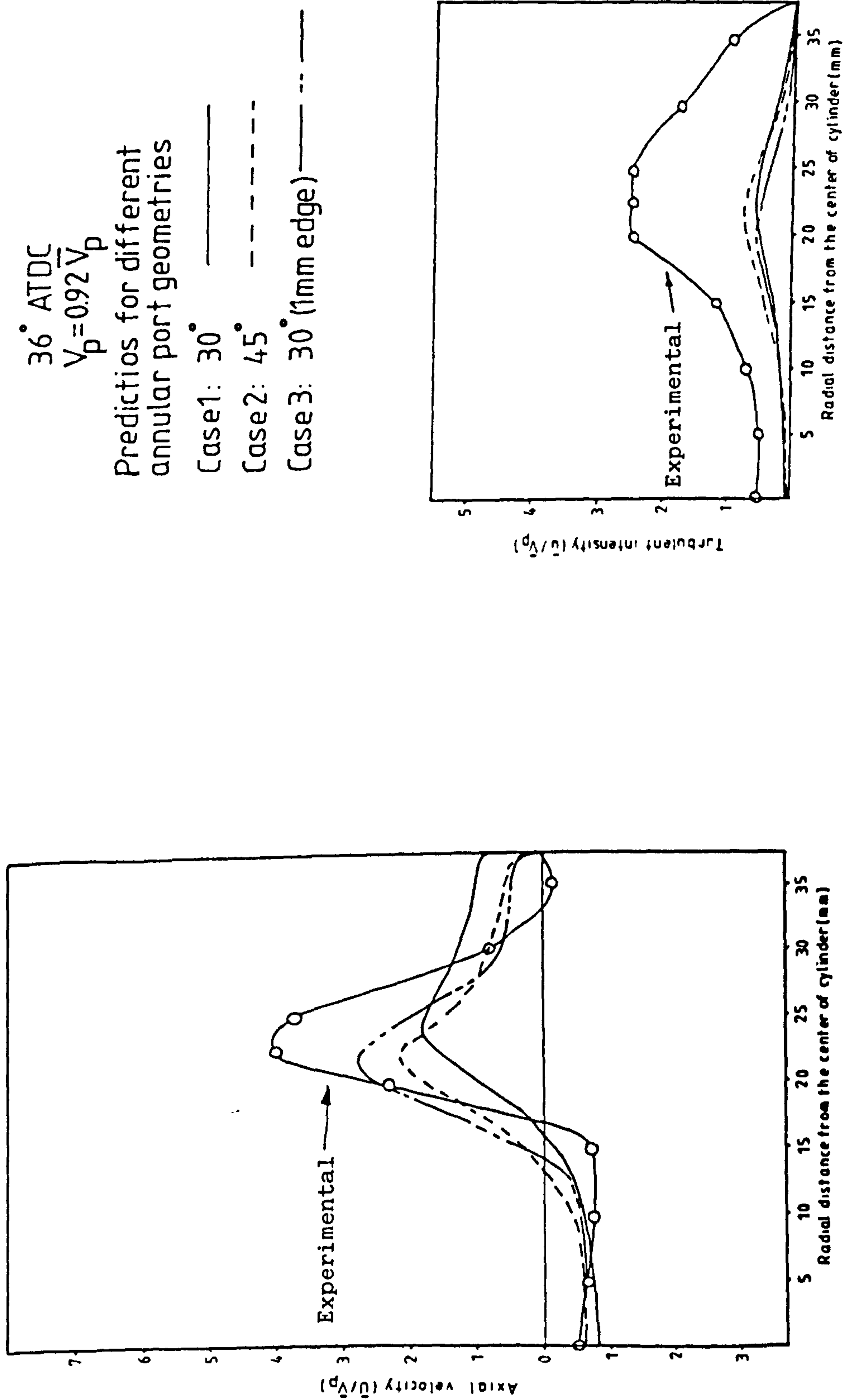
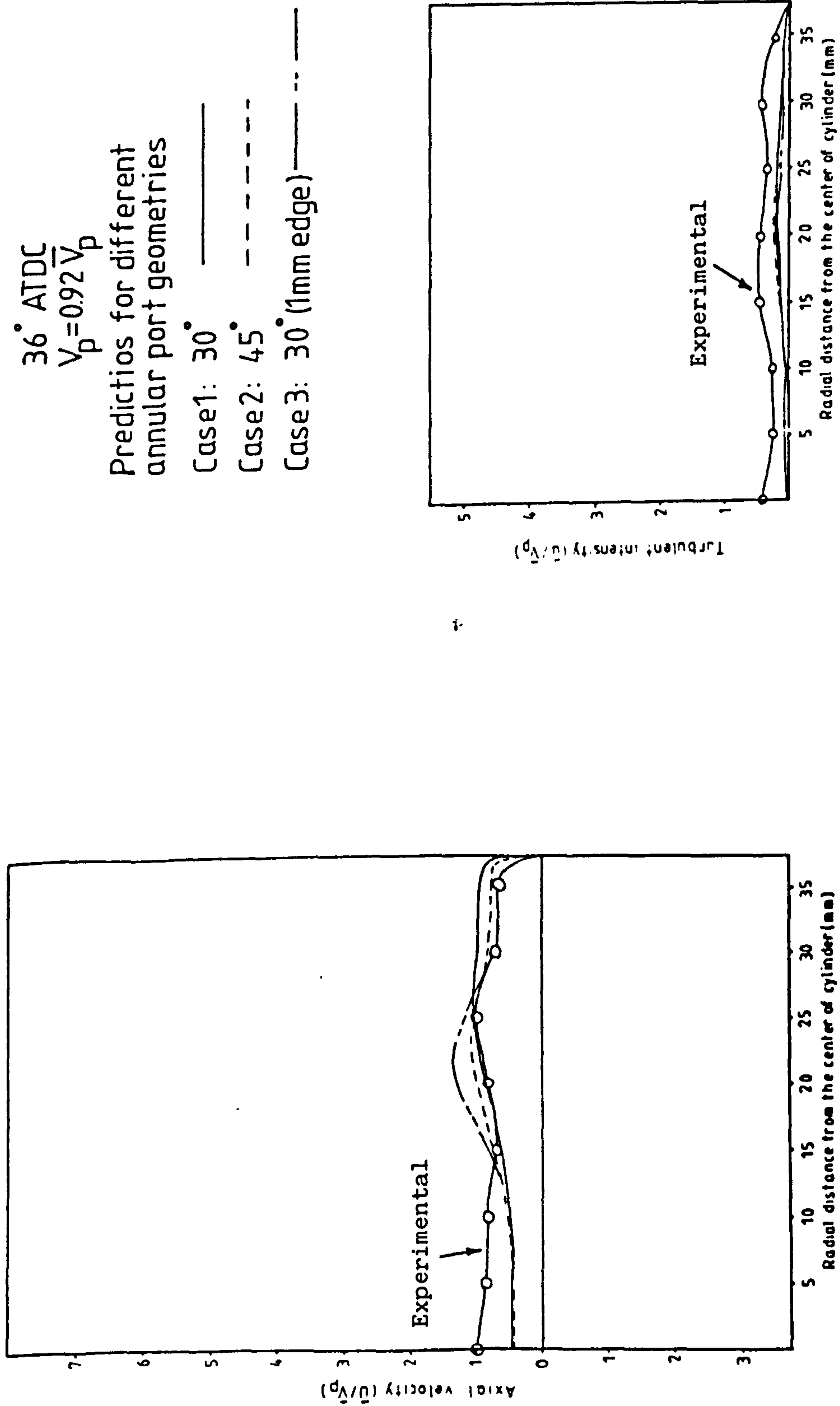


Fig. 7.8 Comparisons between measured and predicted axial velocity and turbulence intensity profiles at 20 mm downstream of the annular valve, for crank angle 36° ATDC.



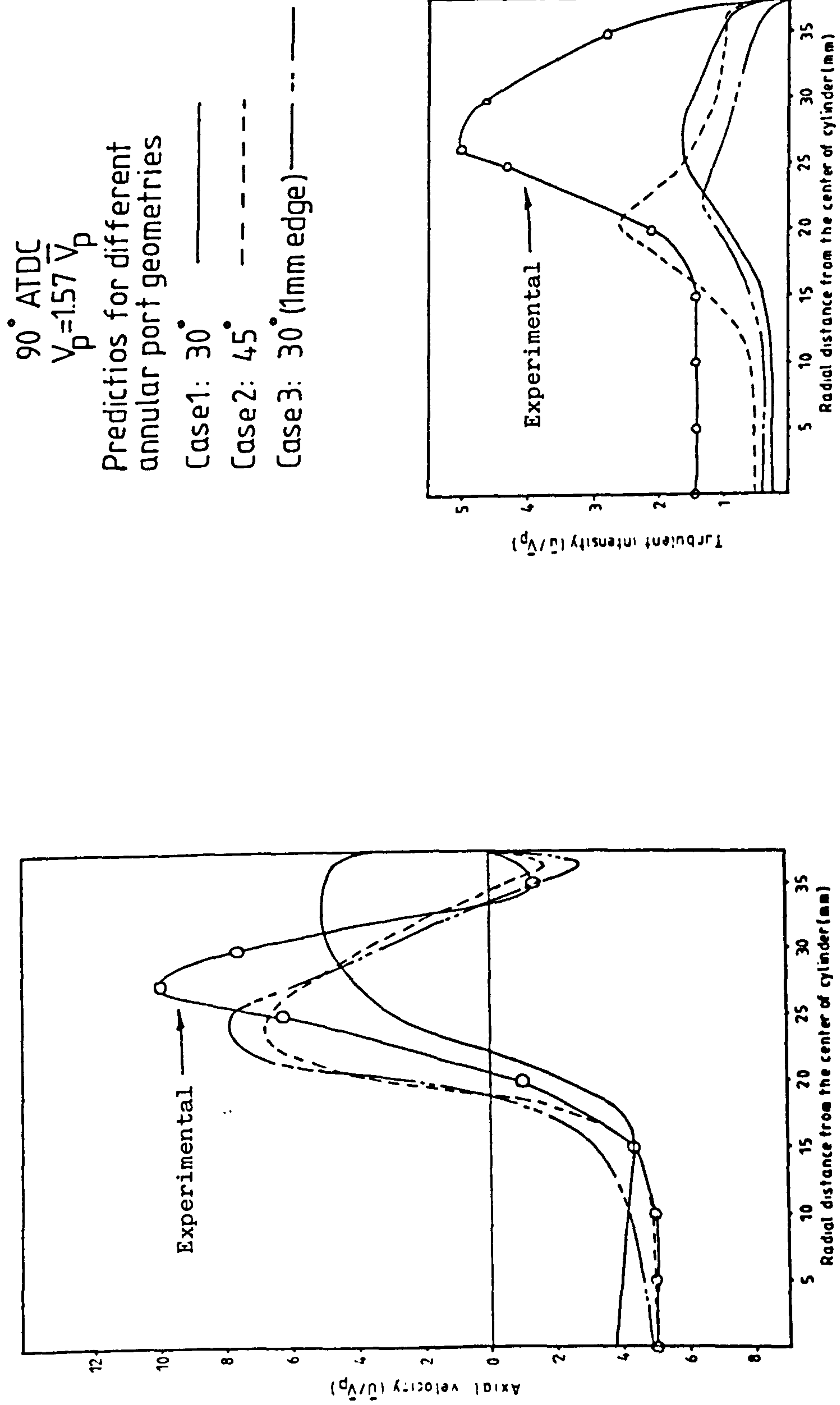


Fig. 7.10 Comparisons between measured and predicted axial velocity and turbulence intensity profiles at 10 mm downstream of the annular valve, for crank angle 90° ATDC.

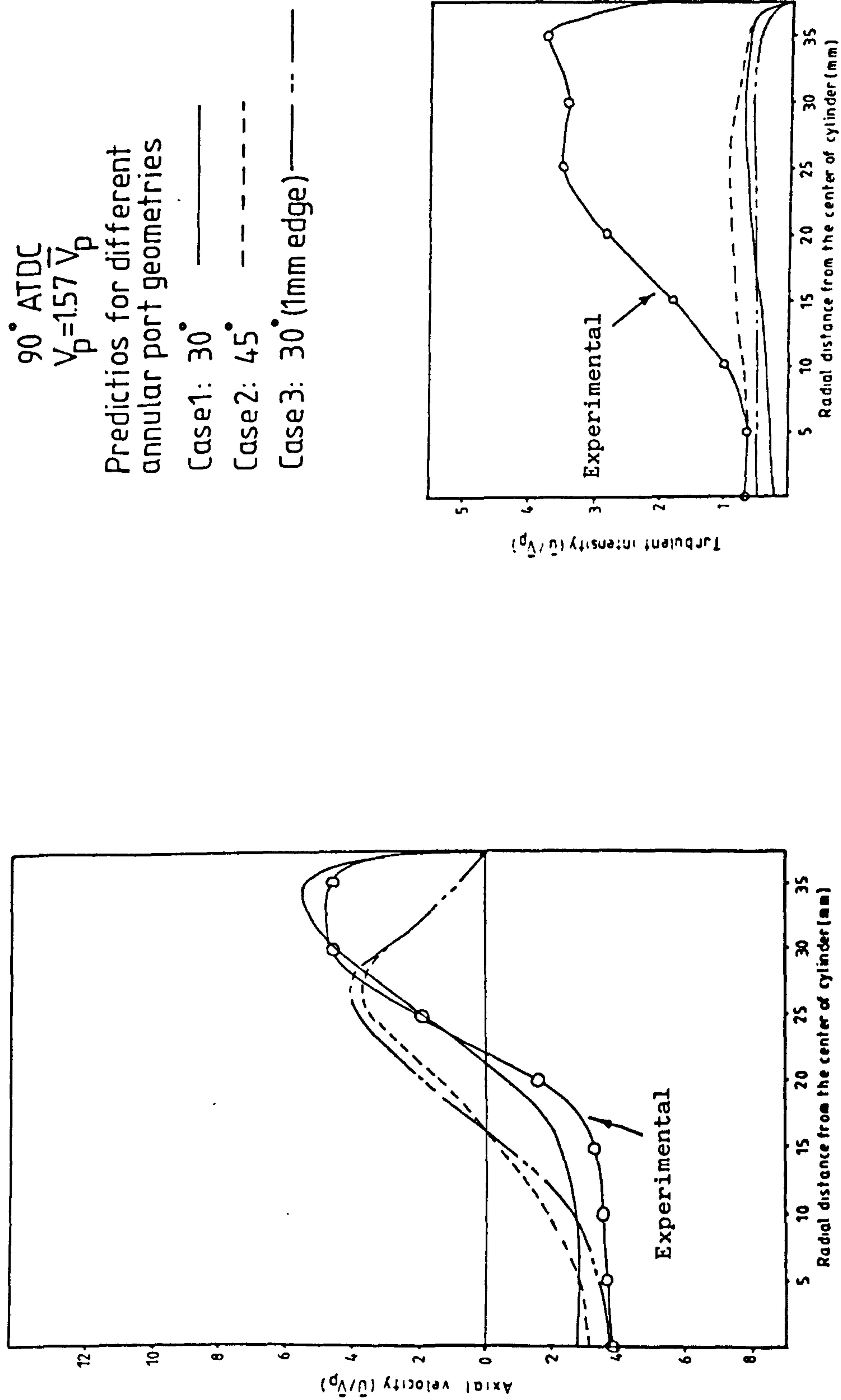


Fig. 7.11 Comparisons between measured and predicted axial velocity and turbulence intensity profiles at 30 mm downstream of the annular valve, for crank angle 90° ATDC.

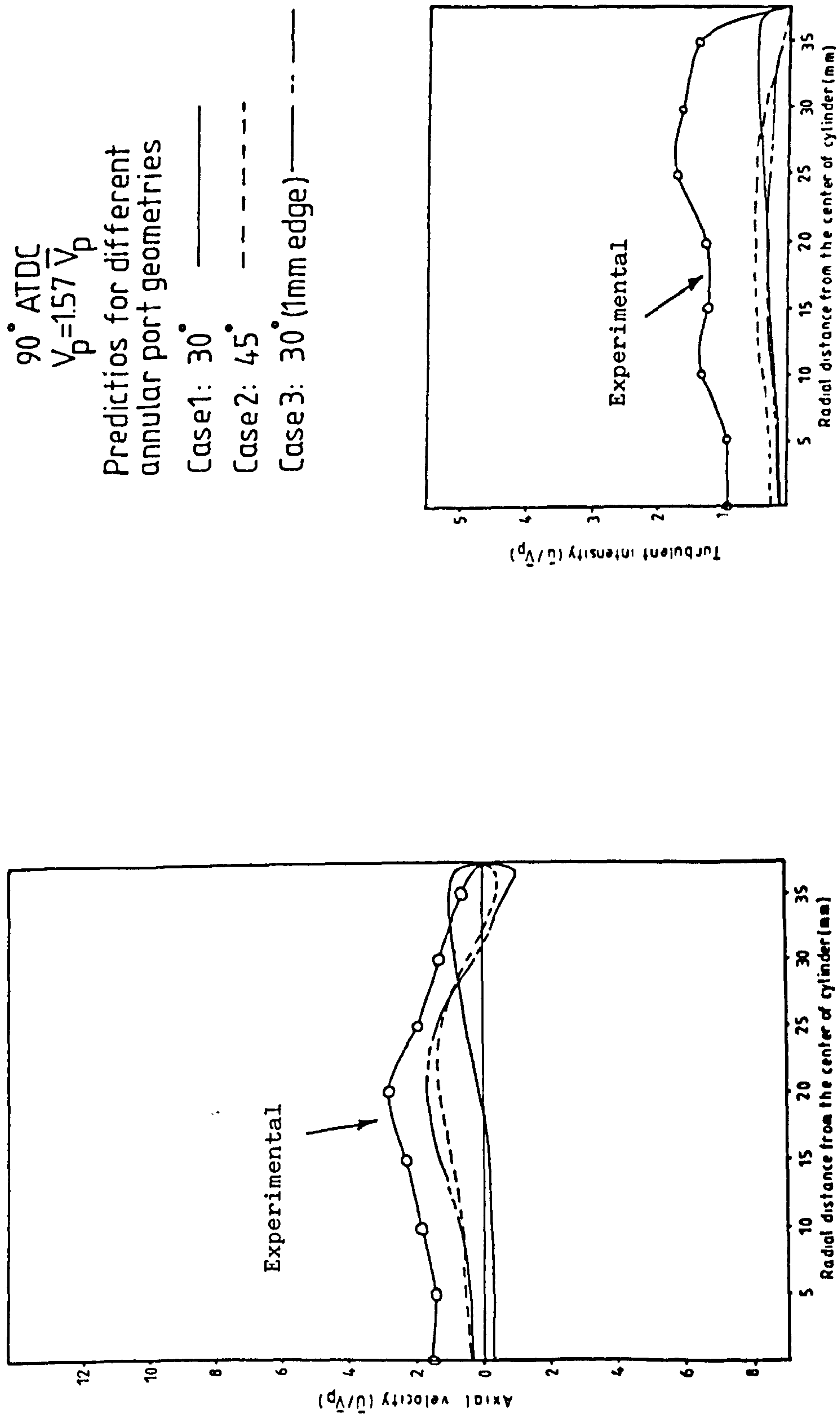


Fig. 7.12 Comparisons between measured and predicted axial velocity and turbulence intensity profiles at 50 mm downstream of the annular valve, for crank angle 90° ATDC.

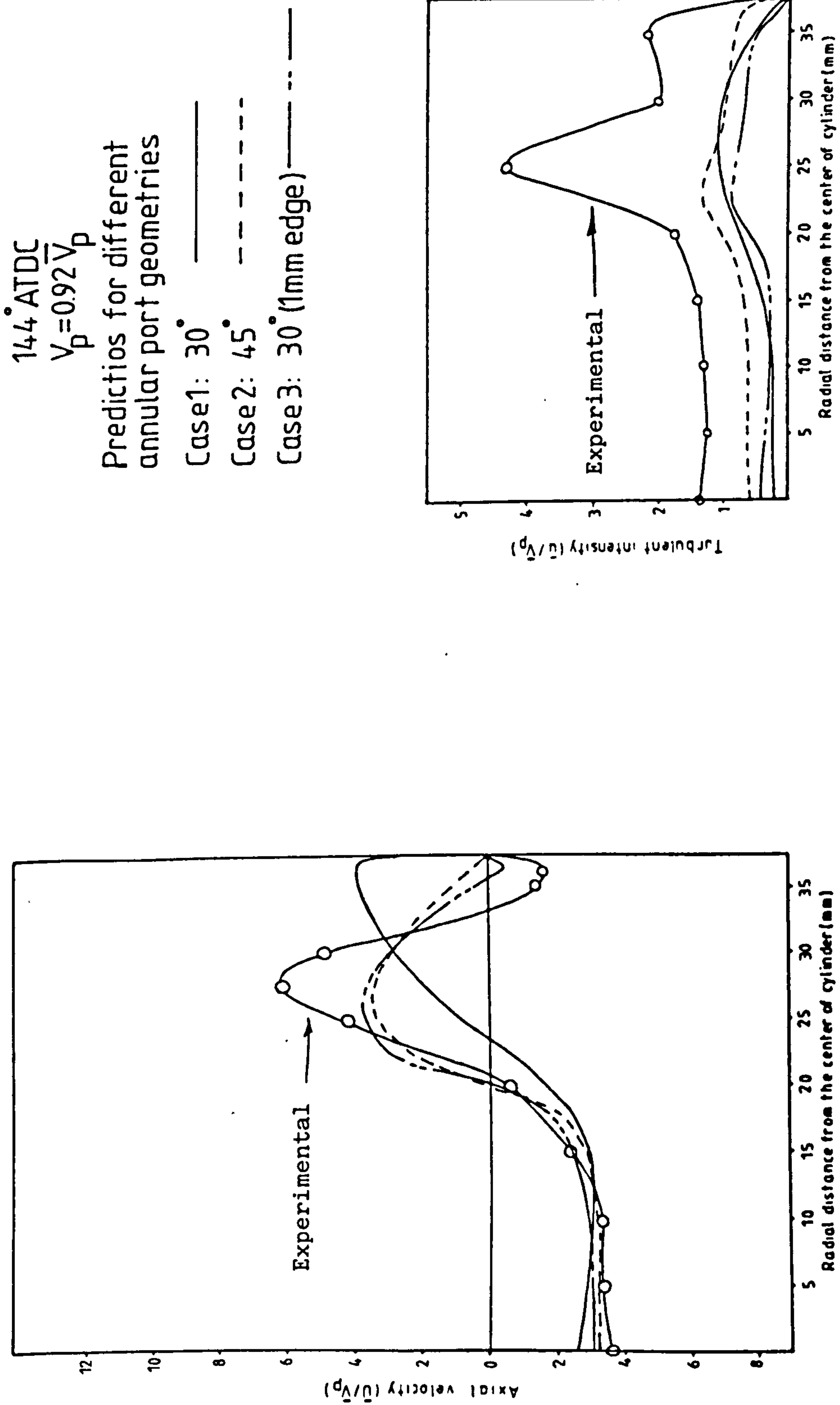


Fig. 7.13 Comparisons between measured and predicted axial velocity and turbulence intensity profiles at 10 mm downstream of the annular valve, for crank angle 144° ATDC.

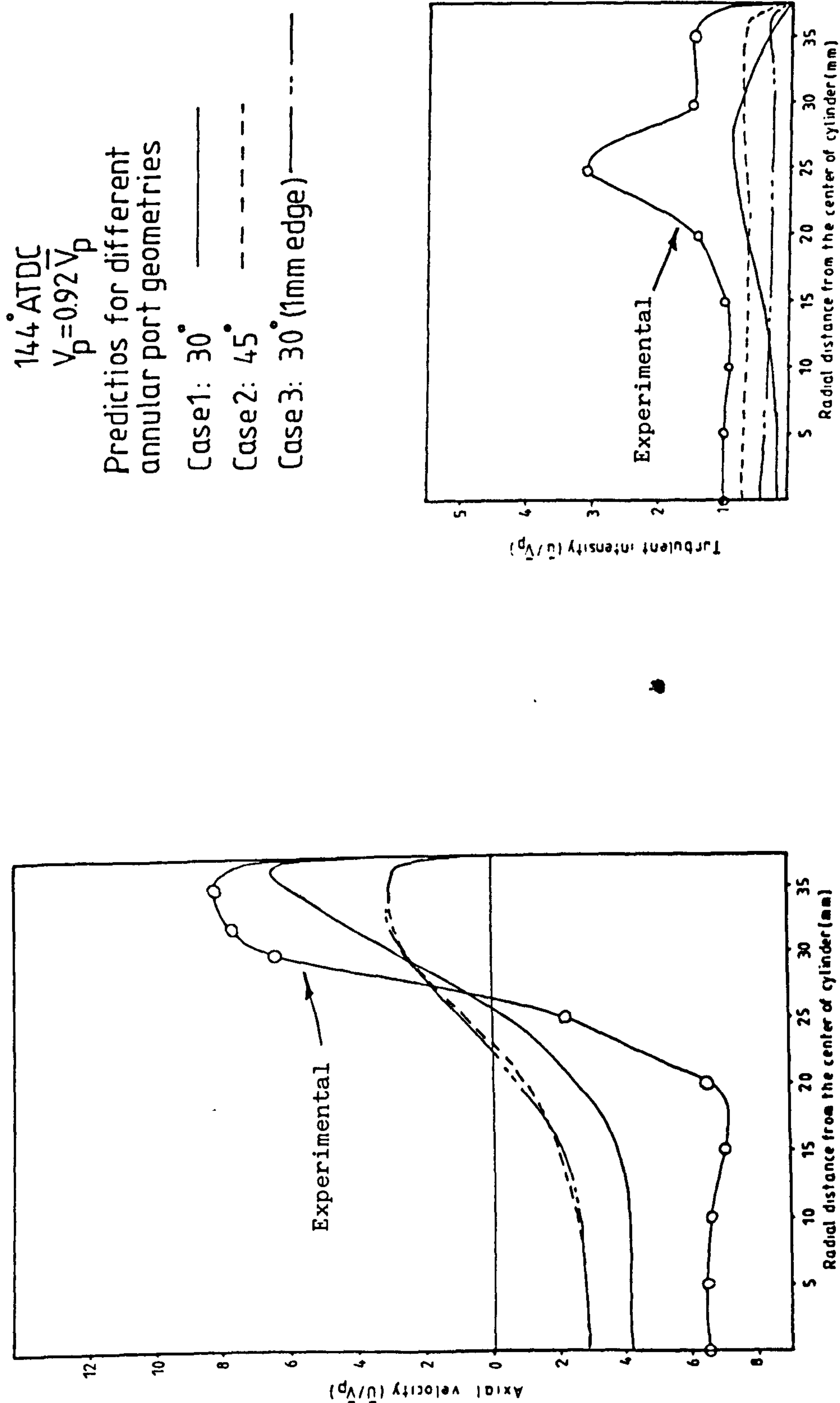


Fig. 7.14 Comparisons between measured and predicted axial velocity and turbulence intensity profiles at 30 mm downstream of the annular valve, for crank angle 144° ATDC.

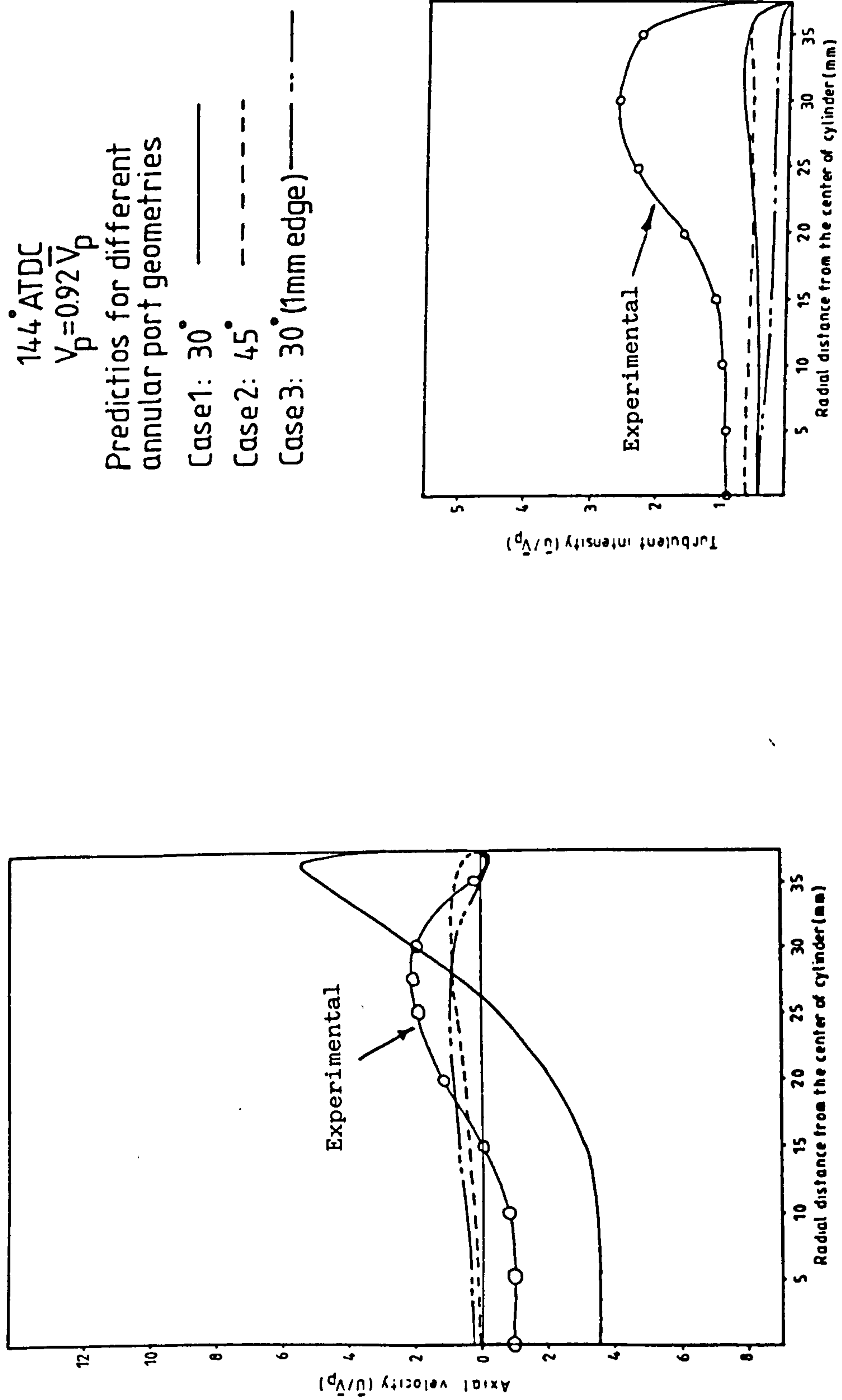


Fig. 7.15 Comparisons between measured and predicted axial velocity and turbulence intensity profiles at 50 mm downstream of the annular valve, for crank angle 144° ATDC.

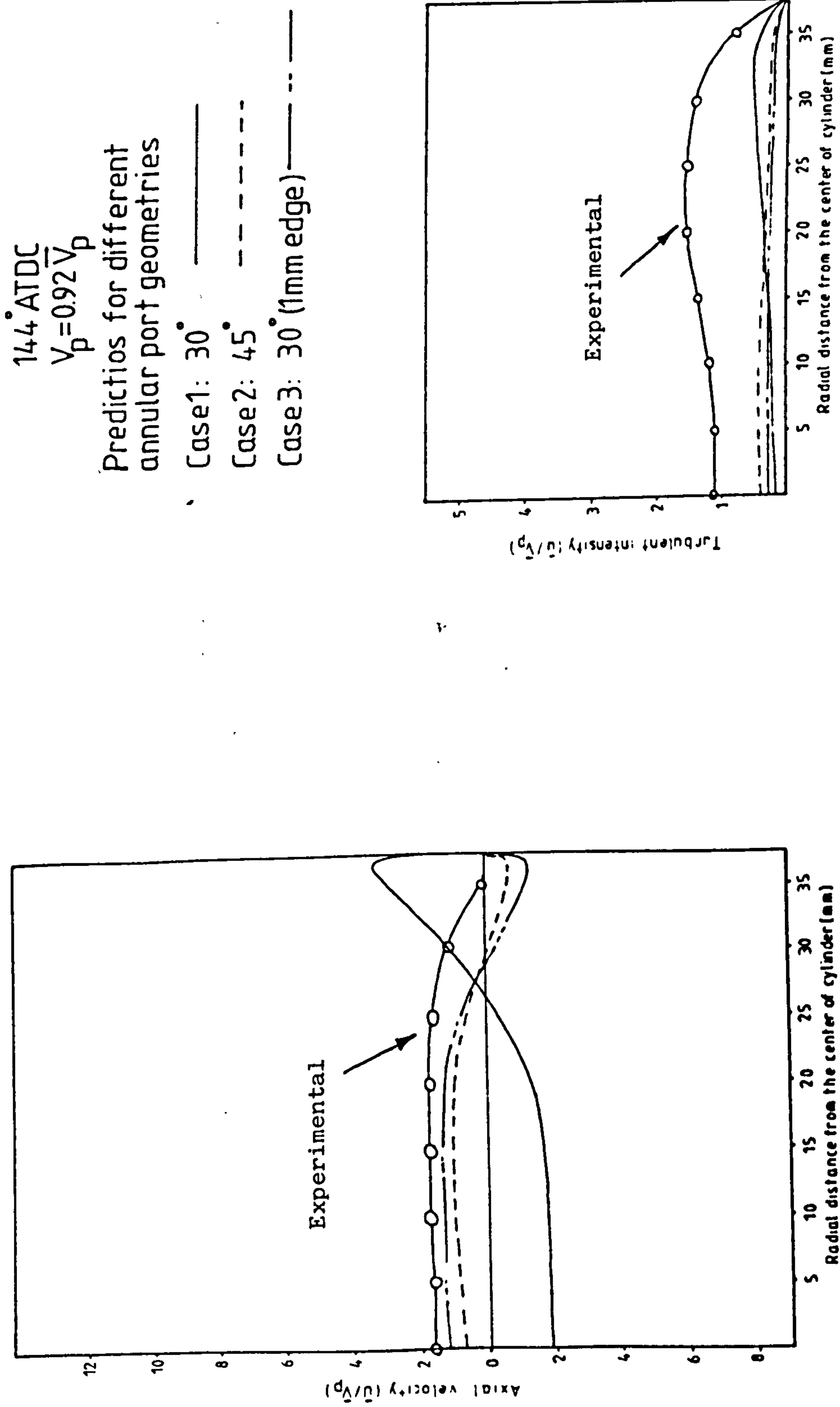


Fig. 7.16 Comparisons between measured and predicted axial velocity and turbulence intensity profiles at 70 mm downstream of the annular valve, for crank angle 144° ATDC.

30° valve (1 mm edge)

45° valve

30° valve

$$C_p = \frac{P_o - P_n}{P_o}$$

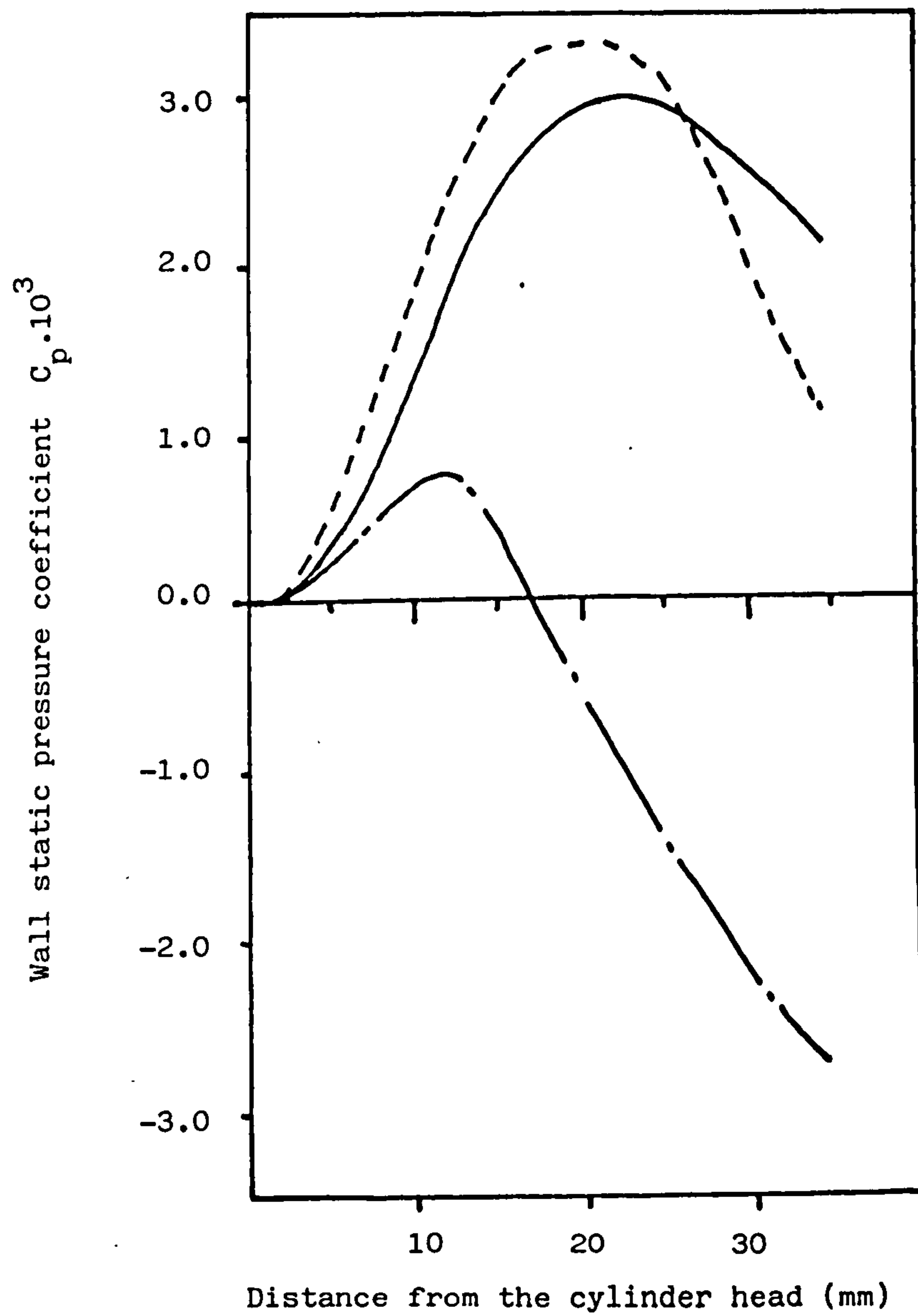


Fig. 7.17 Plot of cylinder wall static pressure coefficient at 36° ATDC

30° valve (1 mm edge)

45° valve

30° valve

$$C_p = \frac{P_o - P_n}{P_o}$$

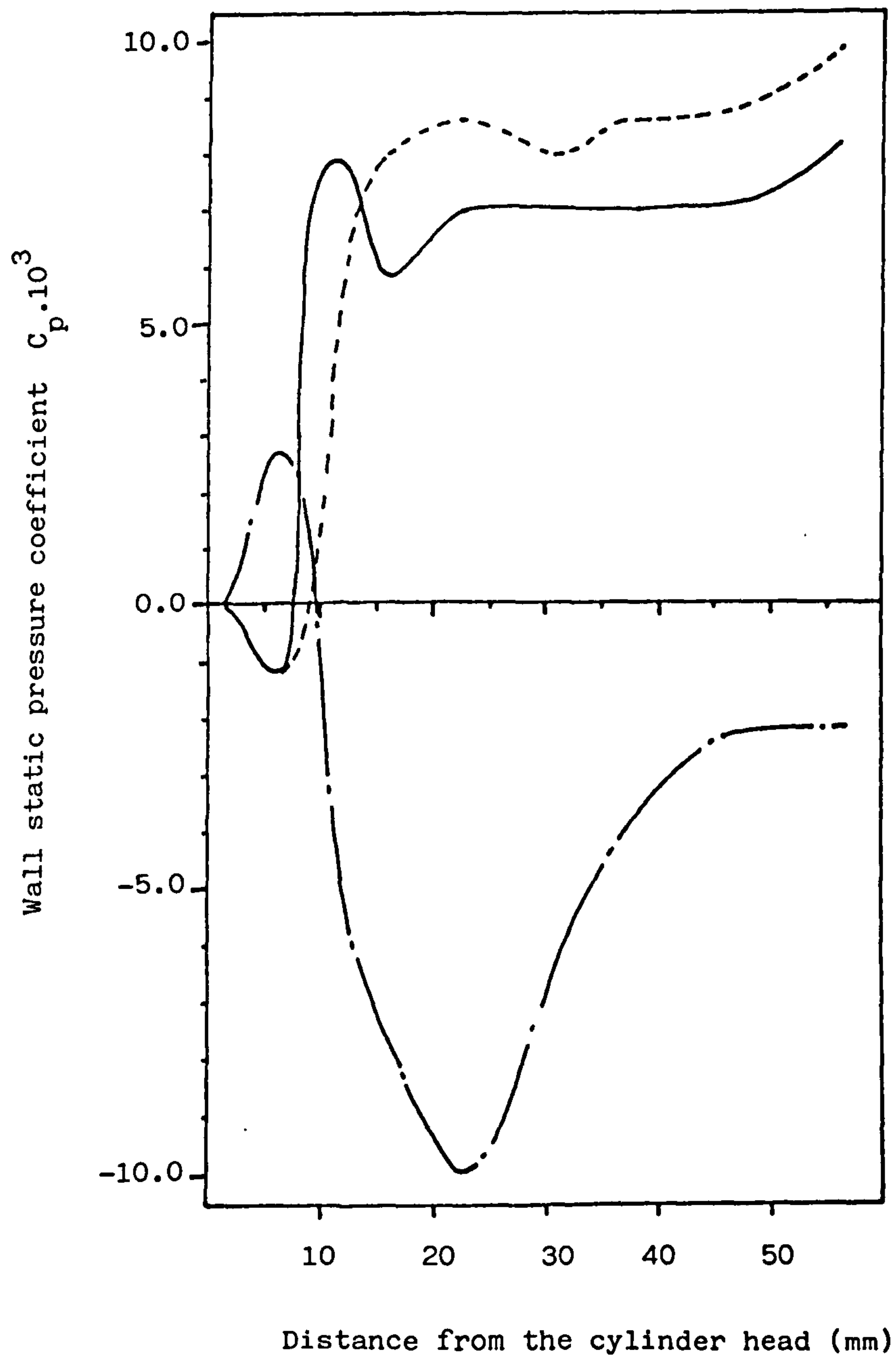


Fig. 7.18 Plot of cylinder wall static pressure coefficient at 90° ATDC

30° valve (1 mm edge)

45° valve

30° valve

$$C_p = \frac{P_o - P_n}{P_o}$$

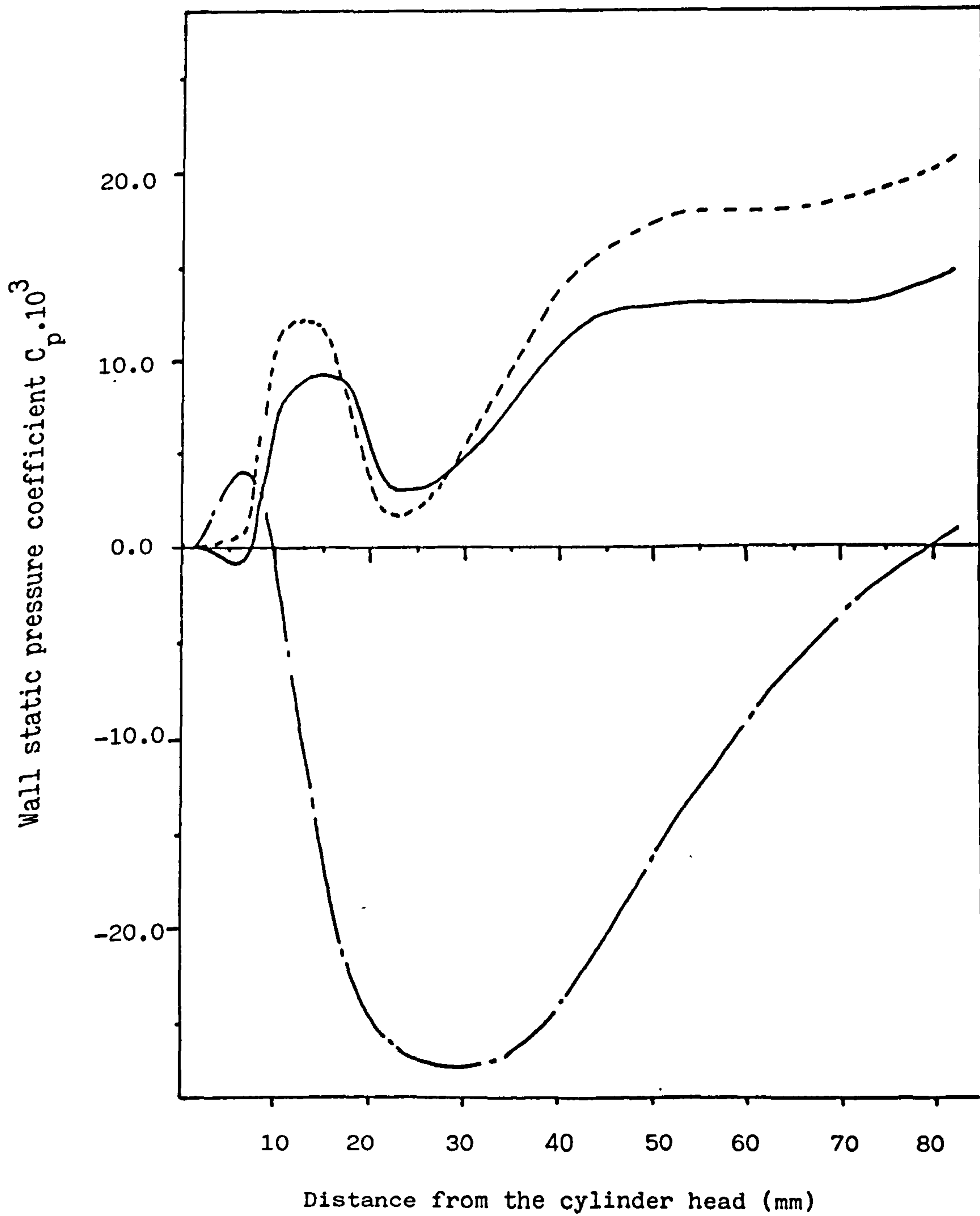


Fig. 7.19 Plot of cylinder wall static pressure coefficient at 144° ATDC

CONCLUDING REMARKS

The main objective and philosophy of the present research in relation to the induction flow and in-cylinder processes of reciprocating engines was to combine computational flow modelling, flow visualization and hot wire point measurements of mean flow and turbulence properties, in order to establish a better, more detailed understanding of the effects of alternative throttling devices on charge preparation and turbulence generation in such systems. In so doing, it also sought to assess the broader diagnostic potential of flow field computational techniques in internal combustion engine design.

The results obtained in Chapter 4 from the water analogy flow visualization tests could be extended only qualitatively to the actual engine. The absence of transient effects and a moving piston made the interpretation of the in-cylinder flow structure difficult, mainly due to the vortex break-up and instability. General features of the flow in the induction tracts and past the throttling devices, however, were found to be stable and substantially independent of flow rate. The photographs of flow tracers in a variety of viewing planes showed that at high throttle settings the turbulence generating ramp, with its steep ramp angle, produce a more pronounced and persistent recirculation zone downstream of the throttle throat, in comparison to the conventional butterfly valve system. In addition, the variable ramp device appeared to provide better flow distribution characteristics by encouraging very little recirculation or diversion of the flow in the secondary induction manifold. At low and wide open throttle (WOT) both systems appeared to show similar mixing characteristics.

In Chapter 5 the largely qualitative data obtained from the water analogy tests were complemented by limited hot wire velocity measurements, with air replacing the water as the flow medium. In view of the inherent limitations in using the hot wire anemometer in the highly turbulent and anisotropic regions of the flow, such as the ones encountered in the cylinder, the comments to some extent were restricted to generalized statements of the trends observed. These measurements, however, were primarily meant to provide an additional and, to some degree, quantitative means of further validation

for the computer predictions. Overall, the agreement between the hot wire and flow visualization results were found to be good. In addition, the flow measurement tests indicated an apparent improvement of volumetric efficiency at low and wide open throttle settings for the variable ramp device, in comparison to the conventional butterfly valve system.

In Chapter 6 the stationary turbulent flow through the alternative induction system throttling devices was modelled computationally using the general purpose PHOENICS code developed by CHAM, and comparisons were made with water analogy flow visualization and hot wire flow measurements. The essential flow field characteristics were satisfactorily reproduced, including in particular the extent of the recirculation zone in the lee of the throttle and the relative persistence of turbulence generated downstream for varying throat apertures. The limited quantitative comparisons with hot wire velocity measurements lend further support to the more detailed aspects of the computational predictions. The predictions did, however, suffer from a number of shortcomings, notably the isotropic viscosity formula assumption in $K \sim \epsilon$ model of turbulence, the absence of partially developed boundary layer and wall friction in partially blocked regions of flow, and the geometric simplifications required in the computational simulations of the flow.

In the context of engine performance it was shown that at high throttle settings the variable ramp throttling system might improve engine performance and the extent of the lean mixture limit of combustion, by improving mixture formation as well as increasing gas motion by both swirl and higher gas velocities past the inlet valve in to the cylinder.

In Chapter 7 comparisons were made between PHOENICS predictions and Laser-Doppler measurements of velocity, for transient turbulent flow fields, in a non-compressing and axisymmetric model engine, reported in the literature. The overall structure of the in-cylinder flow was found to be strongly sensitive to the inlet flow conditions, and in particular to the inlet jet trajectory. The gross features of flow were well predicted depending on the inlet valve seat angle and geometry. Although the mean velocity predictions were relatively well predicted, the shear-generated turbulence intensities were persis-

tently under-predicted. This appeared to be partly due to the lower velocity gradients inside the cylinder and partly to the absence of wall friction or satisfactory turbulence quantities by way of boundary conditions inside the annular inlet port.

The incorporation of appropriate boundary conditions and a more accurate and detailed representation of complex geometries is made possible by the use of a body-fitted coordinate system, now available in PHOENICS but not when the study was initiated, and any future studies should take advantage of this option. Until such times as further studies are undertaken, the net consequences of other defects on overall prediction cannot be fully quantified. In the meantime, the overall agreement between PHOENICS predictions and experimental data was very encouraging, reinforcing the view that general purpose computer codes of the kind investigated can and will play an important role in detailed design assessment and evaluation of internal combustion engines.

REFERENCES

- 1 WITZE, P.O., Hot wire turbulence measurements in a motored internal combustion engine, presented at the 2nd European Symposium on Combustion, Orleans, France, September 5, 1975.
- 2 SEMENOV, E.S., Studies of turbulent gas flow in piston engines, NASA Technical Translation F-97, 1963.
- 3 DENT, J.C. and SALAMA, N.S., The measurement of the turbulence characteristics in an internal combustion engine cylinder, Paper no. C83/75, I. Mech. E. Conference on Combustion in Engines, Cranfield, England, 1975.
- 4 LANCASTER, D.R., Effects of turbulence on spark ignition engine combustion, Ph.D. Thesis, University of Illinois, 1975.
- 5 MEYER, W.E. and BIRNIE, C., Mapping of airflow patterns in engines with induction swirl, S A E Paper no. 660093.
- 6 JAMES, E.H. and LUCAS, G.G., Turbulent flow in spark ignition engine combustion chambers, S A E Paper no. 750885.
- 7 DENT, J.C. and SALAMA, N.S., The measurement of the turbulence characteristics in an internal combustion engine cylinder, S A E Paper no. 750886, Automobile Engineering Meeting, Detroit, October 1975.
- 8 BLIZARD, N.S. and KECK, J.C., Experimental and theoretical investigation of turbulent burning model for internal combustion engines, S A E Paper no. 740191, 1974.
- 9 BALLAL, D.R. and LEFEBVRE, A.H., The influence of flow parameters on minimum ignition energy and quenching distance, 15th Symposium on Combustion, p. 1473, The Combustion Institute, 1975.
- 10 PATTERSON, D.J., Cylinder pressure variations, a fundamental combustion problem. S A E Trans. 75, Paper 660129, 1967.
- 11 MATSUOKA, S., YAMAGUCHI, T. and UMEMURA, Y., Factors influencing the cyclic variation of combustion spark ignition engine, Paper 710586 presented at S A E Mid-Year Meeting, Montreal, June, 1971.
- 12 WINSOR, R.E. and PATTERSON, D.J., Mixture turbulence, a key to cyclic combustion variations, S A E Paper no. 730086, 1973.
- 13 OHIGOSHI, S., HAMAMOTO, Y. and KIZIMA, A., Effects of turbulence on flame propagation in closed vessel, Bull. J.S.M.E. 14, no. 74 (1971).
- 14 LANCASTER, D.R. and KRIEGER, R.B., Effects of turbulence on spark-ignition engine combustion, S A E Paper no. 760160.
- 15 NACA Report 1300, Basic considerations in the combustion of hydrocarbon fuels with air, 1959.
- 16 BEER, J.M. and CHIGIER, N.A., Combustion aerodynamics, John Wiley, New York, 1972.
- 17 SWEET, C.C., Spark ignition of flowing gases, NACA Report no. 1287, June 1956.

- 18 DESOETE, G.G., The influence of isotropic turbulence on the critical ignition energy, 13th Symposium (International) on Combustion, p. 735, The Combustion Institute, 1973.
- 19 KARPOV, V.P. and SOKOLIK, A.S., Ignition limits in turbulent gas mixtures, Institute of Chemical Physics, Academy of Sciences, USSR. Translated from Dokl. Akad. Nauk. SSSR 141 no. 2, 393-396 (1961).
- 20 KALGHATGI, G.T. (1985), Early flame development in a spark ignition engine. Combust. Flame, 60, pp. 299-308.
- 21 MATTHES, W.R. and MCGILL, R.N., Effects of the degree of fuel atomization on single-cylinder engine performance, SAE Paper 760117, SAE Transactions, Vol. 85, 1976.
- 22 DODD, A.E. and WISDOM, J.W., Effect of mixture quality on exhaust emissions from single-cylinder engines. Proceedings of the Institute of Mechanical Engineers, Vol. 183, part 3E, 1968.
- 23 MIZUTANI, Y. and MATSUSHITA, S., Fuel vapor-spray-air mixture operation of a spark-ignition engine. Combustion Science and Technology Vol. 8, 1973, pp. 85-94.
- 24 HARRINGTON, J.A., A study of carburetion effects on power, emissions, lean misfire limit and EGR tolerance of a single-cylinder engine. SAE Paper 760754, presented at the Automobile Engineering Meeting, Dearborn, Michigan, October 18-22, 1976.
- 25 BEALE, N.R. and HODGETTS, D., Inlet valve throttling and the effects of mixture preparation and turbulence on the exhaust gas emissions of a spark ignition engine. Proceedings of the Institute of Mechanical Engineers, Vol. 190 1/76, 1976.
- 26 BEALE, N.R. and HODGETTS, D., The effect of mixture quality on the exhaust gas emissions of a petrol engine. Cranfield Report A.S.A.E. no 1, Cranfield Institute of Technology, Bedford, England, July, 1970.
- 27 LINDSAY, R., THOMAS, A., WOODWORTH, J.A. and ZESCHMANN, E.G., Influence of homogeneous charge on the exhaust emissions of hydrocarbons, carbon monoxide and nitric oxide from a multicylinder engine. SAE Paper 710588, presented at the Mid-Year Meeting, Montreal, Quebec, June 7-11, 1971.
- 28 PETERS, B.D. and QUADER, A.A., Whetting the appetite of spark ignition engines for lean combustion. SAE Paper no. 780234.
- 29 JONES, J.H. and GAGLIARDI, J.C., Vehicle exhaust emissions experiments using a premixed and preheated air fuel charge. SAE Paper no. 670485.
- 30 HUGHES, D.W. and GOULBURN, J.R., Fuel vaporization - economy with reduced exhaust emissions. Proceedings of the I.Mech.E. Vol. 190, pp. 1-76.
- 31 GOULBURN, J.R. and HUGHES, D.W., Mixing of vaporized petrol and air in automobile engine inlet systems. C96/79. Fuel economy and emissions of lean burn engines. London, June 1979, I.Mech.E.
- 32 BEALE, N.R. and HODGETTS, D.H., The effect of mixture quality on the exhaust gas emissions of a petrol engine. Cranfield Report A.S.A.E. no. 1.

- 33 WIATRAK, W. and SLAWIC, J., SZOTT carburettor - a new feeding and control system for lean burn automotive S.I. engines. Fisita XVIII Congress, May 1980, Hamburg.
- 34 KOPA, R.D., Control of automotive exhaust emission by modification of the carburetion system. SAE Paper no. 660114.
- 35 JAMES, E.H., Investigation of turbulent flow in the combustion chamber of a spark ignition engine. Ph.D. Thesis, Loughborough University of Technology, 1972.
- 36 JAMES, E.H. and LUCAS, G.G., Turbulent flow in spark ignition engine combustion chambers. SAE Paper no. 750885.
- 37 WITZE, P.O., Measurement of the spatial distribution and engine speed dependence of turbulent air motion in an I.C. engine. SAE Paper no. 770220
- 38 WITZE, P.O., hot wire measurements of the turbulence structure in a motored spark ignition engine. Sandia Laboratories, 1977.
- 39 NAGAYAMA, I., ARAKI, Y. and IOKA, Y., Effects of swirl and squish on a S.I. engine combustion and emissions. SAE Paper 770217.
- 40 MAYO, J., The effect of engine design parameters on combustion rate in spark ignited engines. SAE Transactions Vol. 84 (1975) Paper 750355.
- 41 LUCAS, G.G. and ANTON, R., The effects of squish on charge turbulence and flame propagation in an S.I. engine. C87/79. Fuel economy and emissions of lean burn engines. London, June 1979, I.Mech.E.
- 42 MA, T.H., Effect of cylinder charge motion on combustion. C81/75. Combustion in engines. Cranfield, July 1975, I.Mech.E.
- 43 JONES, B.E. and MACKWORTH, J.D., Induction swirl in S.I. engines. C85/75. Combustion in engines. Cranfield, July 1975, I.Mech.E.
- 44 LUCAS, G.G., BRUNT, M. and PEDRONIC, S., Lean mixture running of spark ignition engine by generation of a vortex system within the intake. SAE 780964.
- 45 STIVENDER, D.L., Intake valve throttling (ITV), a sonic throttling intake valve engine. SAE 680399.
- 46 HODGETTS, D. and BEALE, N.R., Inlet valve throttling and the effects of mixture preparation and turbulence on the exhaust gas emissions of a spark ignition engine. Proc. I.Mech.E. Vol. 190, 1/76.
- 47 MA, T.H., Effects of cylinder charge motion on combustion. C81/75 Combustion in engines. Cranfield, July 1975, I.Mech.E.
- 48 ABBAS, H.A., Design and performance of a throttling device. M.Sc. Thesis 1982. Cranfield Institute of Technology.
- 49 ABDULLA, G.A., The performance of turbulence generating and throttling device. M.Sc. Thesis 1983. Cranfield Institute of Technology.
- 50 SANATIAN, R., Models of the flow in the induction tract of a reciprocating engine. M.Sc. Thesis 1984. Cranfield Institute of Technology.

- 51 EKCHIAN, A. and HOULT, D.P., Flow visualization study of the intake process of an internal combustion engine. SAE Paper 790095.
- 52 NICHOLSON, H.M. and FIELDS, J.P., Some experimental techniques to investigate the mechanism of flame stabilization in the wakes of bluff bodies. Third Symposium on Combustion, Flame and Explosion Phenomena. Williams & Williams, Baltimore, 1949.
- 53 GERRARD, A.J. and McAREAREG, G., The use of models in design and development, Birmingham University Chemical Engineer 8, 2 (1957).
- 54 YOUNG, G.A. and MILLAR, R.J.O., Some considerations on the use of water models in study of air flow problems. British Hydromechanics Research Association Report no. SP.461 (1953).
- 55 MELLING, A. and WHITELOW, J.H., Seeding of gas flows for laser anemometry. DISA Information 15, 5-14 (1973).
- 56 DURST, F., MELLING, A. and WHITELOW, J.H., Principles and practice of laser-doppler anemometry. Academic Press, London, 2nd edition (1981).
- 57 MELLING, A., Seeding gas flows for laser anemometry. AGARD-CP-399. Paper 8 (1986).
- 58 COLLIS, D.C. and WILLIAMS, M.J., Two-dimensional convection from heated wires at low Reynolds numbers. J. Fluid Mech. 6, 357 (1959).
- 59 EMRICH, R.J., Methods of experimental Physics. Volume 18 Fluid Dynamics. Part A. Academic Press, New York (1981).
- 60 CORRSIN, S., Turbulence: experimental methods. Handbuch der Physik, Vol 8, Part 2. Springer, Berlin, 1963.
- 61 BOURKE, P.J., PULLING, D.J., GRILL, L.E. and DENTON, W.H., The measurement of turbulent velocity fluctuations and turbulent temperature fluctuations in the supercritical region by a hot-wire anemometer and a "cold" wire resistance thermometer. Proc. Inst. Mech. Engrs. 182, Part 31 (1967-8).
- 62 SCHUBAUER, G.B., Advances in hot-wire anemometry (W.C. Melnik and J.R. Wesk, eds.), P 13. University of Maryland, College Park, Md. 1968.
- 63 BENSON, R.S. and BARUAH, P.C., Non-steady flow through a gauze in a duct. J. Mech. Engng. Sci. 1965 7 (no. 4).
- 64 BENSON, R.S., BARUAH, P.C., SIERENS, I.R., Steady and non-steady flow in a simple carburettor. Proc. Inst. Mech. Engrs. Vol 188 53 (1974).
- 65 PHOENICS Instruction Manual. Spring 1983 CHAM TR/75
- 66 SPALDING, D.B. (1982), Four lectures on the PHOENICS computer code. Fluid Dynamics Unit Report Number CFD/82/5.
- 67 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 Mass Transfer 15, 1787.
- 68 REYNOLDS, W.C., Computation of turbulent flows. Ann. Rev. Fluid Mech., Vol. 8, pp. 183-208, 1976.

- 69 BRADSHAW, P., ed., Turbulence. Spring Verlag, 1976.
- 70 BOUSSINESQ, J. (1877), Theorie de l'écoulement tourbillant. Mem. Pre. par. div. Sav. 23, Paris.
- 71 PRANDTL, L. (1925), Bericht über Untersuchungen zur ausgebildeten Turbulenz. ZAMM 5, 136.
- 72 PRANDTL, L., Über ein neues Formalsystem für die ausgebildete Turbulenz, Nachrichten von der Akad. der Wissenschaft in Gottingen (1945).
- 73 BRADSHAW, P., FERRISS, D.H. and ATWELL, N.P., Calculation of boundary-layer development using the turbulent energy equation, J. Fluid Mechanics 28 (1967) p. 593.
- 74 NEE, V.W. and KOVASZNAVY, L.S.G., The calculation of the incompressible turbulent boundary layer by a simple theory. Proc. of AFOSR/IFP Conference on Computation of Turbulent Boundary Layers Vol. I, Stanford University (1968), also Physics of Fluids 12 (1969) p. 473.
- 75 KOLMOGOROV, A.N., Equations of turbulent motion of an incompressible turbulent fluid, Izv. Akad. Nauk SSSR Ser. Phys. VI, no. 1-2 (1942), p. 56.
- 76 SPALDING, D.B., The prediction of two-dimensional, steady turbulent flows, Imperial College, heat Transfer Section Report EF/TN/A/16 (1969).
- 77 JONES, W.P. and LAUNDER, B.E., The prediction of laminarization with a 2-equation model of turbulence. International J. Heat and Mass Transfer 15 (1972) p. 301.
- 78 HANJALIC, K. and LAUNDER, B.E., A Reynolds-stress model of turbulence and its application to asymmetric shear flows. J. Fluid Mechanics 52 (1972) p. 609. (Also available as Imperial College Heat Transfer Section Report TM/TN/A/8).
- 79 LAUNDER, B.E. and SPALDING, D.B., The numerical computation of turbulent flows. Computer Methods in Applied Mathematics and Engineering 3 (1974) 269-289.
- 80 SPALDING, D.B., Monograph on turbulent boundary layers. Imperial College. Mechanical Engineering Department Report TWF/TN/33 (1967).
- 81 WOLFSHTEIN, M.W., The velocity and temperature distribution in one dimensional flow with turbulence augmentation and pressure gradient. International J. Heat and Mass Transfer 12 (1969) p. 301.
- 82 PATANKAR, S.V. and SPALDING, D.B., Heat and mass transfer in boundary layers. 2nd Edition (Intertext Books, London, 1970).
- 83 LAUNDER, B.E. and SPALDING, D.B., Mathematical models of turbulence. Academic Press, 1972.
- 84 JONES, W.P. and LAUNDER, B.E., The calculation of low Reynolds number phenomena with a two-equation model of turbulence. Int. J. Heat and Mass Transfer Vol. 16 pp. 1119-1130, 1973.

- 85 MORSE, A.P., WHITELAW, J.H. and YIANNESKIS, M., Turbulent flow measurements by laser-doppler anemometry in motored piston cylinder assemblies. *Journal of Fluids Engineering*, Vol. 101, June 1979, pp. 208-216.
- 86 GOSMAN, A.d., JOHNS, J.R. and WATKINS, A.P., Development of prediction methods for in-cylinder processes in reciprocating engines. General Motors Research Laboratories Symposium on Combustion Modelling in Reciprocating Engines. Warren, Michigan, November 1978, pp. 69-129.
- 87 HORLOCK, J.H. and WINTERBONE, D.S. (eds.) (1986), *The thermodynamics and gas dynamics of internal combustion engines*. Clarendon Press.

APPENDIX ATHE STRUCTURES OF SATELLITE AND GROUND STATIONA1 General

As mentioned earlier, PHOENICS consists of three main sections: SATELLITE, GROUND and EARTH. The purpose of this Appendix is to help the reader in understanding the steps which have to be taken in constructing computer models of the flow.

The information to be supplied via SATELLITE and GROUND is organised in 43 groups in the main program of SATELLITE SATLIT and 16 chapters of GROUND.

A2 The 43 DATA groups of SATELLITE SATLIT

Information about the type of flow (i.e. parabollic, elliptic), time dependency of the flow, the nature of coordinate system (i.e. Cartesian, Polar), the overall dimensions of the integration domain, and the number and distribution of the sub-dimensions is set in groups 1 to 5.

The EARTH program contains a number of moving-grid options in which the grid is expanded or contracted in z-direction. The information on expansion and contraction of grid is specified in group 6.

Representation of blockages in the flow domain can be made by prescription of porosities in group 7.

Groups 8 to 13 mainly deal with selection of the variables to be solved, fluid properties (i.e. density, viscosity, enthalpies, etc.), interphase processes, special sources and initial fields.

Boundary and internal conditions are specified in groups 14 to 24, selection of regions to be considered are made in group 14 and specific prescriptions for regions 1 to 10 are made in groups 15 to 24.

Solution type and related parameters, sweep and iteration number, termination criteria on sweep and iteration loops and relaxation

factors are specified in groups 25 to 33.

Groups 34 to 39 deal primarily with printout of variables and their error and residual.

The remaining four SATELLITE groups deal with special data, multiruns, restarting and arrangement for graphic printout of the results.

A3 GROUND connecting sub-routines and chapters

There are three connecting sub-routines which provide access to variables in EARTH. They are as follows:

- Sub-routine GET. This is used to get variables from EARTH and store them in local arrays for use in GROUND.

- Sub-routine SET. This is used to prescribe values of variables in EARTH. Whatever was already stored in EARTH array will be overwritten by whatever is prescribed from GROUND.

- Sub-routine ADD. This is used to add additional sources to the source terms for any of the solved for variables, including the introduction of sources to represent non-linear boundary conditions.

GROUND is sub-divided into 16 chapters, called at particular stages in the EARTH solution sequence according to prescriptions in SATLIT.

Chapters 0 and 1 are visited at the start of each run and each time step respectively.

Chapter 2 is called at the start of each sweep of iterative solution, and chapters 3 and 4 are called just before the solutions start at each z slab and at the start of each hydrodynamic iteration performed at each z slab.

Any source-term modifications are made in chapter 5, which is called during solution for each variable at the z slab in question.

Chapter 6 is called at the end of each hydrodynamic iteration loops at a slab, and chapter 7 is called at the end of operations at a slab, just before solution moves to the next slab.

Chapter 8 is only visited for elliptic flows and it occurs

at the end of each sweep of iterative solution. Chapter 9 is called at the end of each time step, for unsteady flows.

Chapter 10 to 16 of GROUND perform different functions from those discussed so far. They are provided specifically for the adjustment of material properties or auxiliary quantities such as density, viscosity, exchange coefficient, etc.

This Appendix provided the briefest possible commentary on the grouping and structure of SATELLITE and GROUND. The reader should refer to PHOENICS Instruction Manual for a more comprehensive information.