

FILM COOLING OF GAS TURBINE

BLADES

by

Adrian James White, B.Tech.

Thesis submitted for the degree of
Doctor of Philosophy,
in the Faculty of Engineering,
University of London.

1982.

ABSTRACT

The research reported in this thesis is concerned with the aerodynamic and heat transfer characteristics of film cooling from configurations of relevance to gas turbine blades.

A five-hole probe is used to measure the velocity field arising from the ejection of cooling air on a simulated gas turbine blade leading edge; and the scalar field measured by a helium trace concentration technique. Ejection from a single row of holes along the stagnation line is unlike that of jets in crossflow, and the flow becomes bistable for jet to freestream velocity ratios, $U > 1$, however, ejection from rows of holes at other locations displays similarities with jets in crossflow.

Two numerical procedures based on finite difference representations of the time averaged equations of mass and momentum conservation, and the two equation $k-\epsilon$ turbulence model are described. The methods are adapted and applied to the film cooling from angled slots, a single normal hole and at a leading edge for $0.24 \leq U \leq 2.3$.

The pressure field generated by the interaction between the ejected flow and the mainstream causes a distortion of the flow within the ejection orifice. The solutions for the two dimensional slot are found to be sensitive to the distortion and a simple model proposed to account for some of its features. A more rigorous treatment is developed for the single hole in which the numerical procedure is employed to calculate the flow within the hole.

A system of embedded solutions is constructed for the computation of film cooling at a leading edge which provides flexibility in the description of the flowfield resulting in significant improvements in the accuracy of the solution.

In the main the predictions for the three configurations correspond with experimental measurements, but certain discrepancies are found which are variously attributed to inadequacies in the turbulence modelling and numerical error.

ACKNOWLEDGEMENTS

I would like to express my gratitude to my supervisors, Professor J.H. Whitelaw and Dr. A.E. Forest for their guidance and advice throughout the research, and to Dr. A.D. Gosman for the contribution of the TEACH program and advice on the numerical work.

I would also like to thank Rolls-Royce Ltd. for allowing me to complete the research and in particular Mr. H. Tubbs and Mr. B. Barry whose support made this project possible.

The encouragement of my colleagues at Rolls-Royce Ltd. was much appreciated, and the advice of Drs. W.P. Jones and C.H. Priddin who provided the three-dimensional numerical method has been invaluable.

The help of the technical staff and students of Imperial College contributed greatly to the completion of the experimental work and in particular I would like to thank Mr. N. Fox, Mr. F. Bell, Mr. B.E. Thompson and Mr. Z. Khan.

I am grateful to Mrs. J. Woolvin whose typing skills have resulted in the final presentation of this thesis and to Mr. R.J. Beacock for drawing the figures.

CONTENTS

	<u>Page</u>
ABSTRACT	1
ACKNOWLEDGEMENTS	3
CONTENTS	4
<u>CHAPTER 1</u>	10
<u>INTRODUCTION</u>	
1.1 Preliminary Remarks	10
1.2 Objectives of the present study	13
1.3 Discussion of film cooling data	14
1.3.1 Two dimensional film cooling	15
1.3.1.1 Adiabatic wall effectiveness	16
1.3.1.2 Heat transfer coefficient	17
1.3.1.3 Density ratio	17
1.3.1.4 Injection geometry	18
1.3.1.5 Upstream boundary layer	19
1.3.1.6 Mainstream turbulence	19
1.3.1.7 Pressure gradient	20
1.3.1.8 Surface curvature	20
1.3.1.9 Compressibility	21
1.3.2 Three dimensional film cooling	21
1.3.2.1 Adiabatic wall effectiveness	22
1.3.2.2 Heat transfer coefficient	22
1.3.2.3 Density ratio	23
1.3.2.4 Injection geometry	24
1.3.2.5 Upstream boundary layer	26
1.3.2.6 Mainstream turbulence	27
1.3.2.7 Pressure gradient	28

	<u>Page</u>
1.3.2.8 Surface curvature	29
1.3.2.9 Compressibility	29
1.3.2.10 Blade environment	30
1.3.3 Film Cooling at a leading edge	31
1.4 Theoretical models of the film cooling process	33
1.4.1 Two dimensional models	33
1.4.2 Three dimensional models	35
1.4.3 Film cooling on curved surfaces	39
1.5 Outline of the thesis	39
 <u>CHAPTER 2</u>	 40
<u>APPARATUS AND MEASUREMENT TECHNIQUES</u>	
2.1 Introduction	40
2.2 Apparatus	41
2.2.1 Windtunnel	41
2.2.2 Test piece	41
2.2.3 Secondary air supply	42
2.2.4 Traverse arrangements	43
2.2.5 Measurement probes	44
2.3 Experimental conditions	45
2.4 Measurement techniques and operating procedures	48
2.4.1 Flow visualisations	48
2.4.2 Static pressures	49
2.4.3 Velocities	49
2.4.4 Concentration	52
2.5 Accuracy of the measurements	53
2.5.1 Static pressure	53
2.5.2 Velocity	54
2.5.3 Concentration	58

	<u>Page</u>
<u>CHAPTER 3</u>	60
<u>PRESENTATION AND DISCUSSION OF THE MEASUREMENTS</u>	
3.1 Introduction	60
3.2 Results without secondary air ejection	60
3.3 Single row; $\Theta_{\gamma} = 0$ degrees	61
3.3.1 Flow visualisations	61
3.3.2 Static pressure distribution	63
3.3.3 Velocity profiles	63
3.3.4 Concentration measurements	64
3.4 Two rows; $\Theta_{\gamma} = \pm 20$ degrees	65
3.4.1 Flow visualisations	65
3.4.2 Velocity profiles	66
3.4.3 Concentration measurements	68
3.5 Two rows; $\Theta_{\gamma} = \pm 40$ degrees	69
3.5.1 Flow visualisations	69
3.5.2 Velocity profiles	71
3.5.3 Concentration measurements	71
3.6 Concluding remarks	72
 <u>CHAPTER 4</u>	 75
<u>THEORETICAL CONSIDERATIONS</u>	
4.1 The governing equations	75
4.2 The equations of motion for turbulent flow	77
4.3 Turbulence modelling	79
4.3.1 Choice of turbulence model	79
4.3.2 The $k-\epsilon$ turbulence model	81
4.4 Numerical solution of the equations	83
4.4.1 The finite difference equations	83
4.4.2 Calculation of the pressure field	86
4.4.3 Solution of the equations	88

	<u>Page</u>
4.5 Boundary conditions	90
4.5.1 Pressure correction equation	90
4.5.2 Boundary conditions at a wall	91
<u>CHAPTER 5</u>	96
<u>TWO DIMENSIONAL FLUSH SLOT CALCULATIONS</u>	
5.1 Introduction	96
5.2 Computational details	96
5.2.1 Boundary conditions	96
5.2.2 Determination of the size of the solution domain	102
5.2.3 Numerical accuracy	103
5.2.4 Turbulence modelling	109
5.3 Comparison between prediction and measurement	111
5.3.1 Ninety degree slots	111
5.3.2 Sixty degree slots	113
5.3.3 Thirty degree slots	114
5.4 Concluding remarks	115
<u>CHAPTER 6</u>	117
<u>JET IN CROSSFLOW CALCULATIONS</u>	
6.1 Introduction	117
6.2 Computational details of the solution for the jet-mainstream interaction	119
6.2.1 Boundary definition	119
6.2.2 Numerical accuracy	122
6.2.3 Continuation downstream	123
6.3 Computational details of the solution within the hole	124
6.3.1 The need for computation within the hole	124
6.3.2 Boundary definition	125
6.3.3 Numerical accuracy	127

	<u>Page</u>
6.3.4 Linkage with the solution of the jet-mainstream interaction	127
6.4 Comparison with experimental measurements	130
6.4.1 Velocity profiles	130
6.4.2 Effectiveness contours	132
6.4.3 Surface effectiveness	133
6.4.4 Surface heat transfer	135
6.5 Concluding remarks	136
<u>CHAPTER 7</u>	138
<u>LEADING EDGE FILM COOLING CALCULATIONS</u>	
7.1 Introduction	138
7.2 Computational details	140
7.2.1 Treatment of the rectangular grid cells at the interface	140
7.2.2 Determination of boundary values at the interface for the cylindrical grid	143
7.2.3 Validation of the rectangular grid solution	145
7.2.4 Validation of the embedded grid solutions for a cylinder without ejection	147
7.2.5 Validation of the cylindrical grid solution with ejection	150
7.3 Comparison with experimental measurements	152
7.3.1 Measurements of Rastogi and Whitelaw (1972); $\Theta_s = 0$ degrees	152
7.3.2 Measurements of the present study	154
7.4 Concluding remarks	156

	<u>Page</u>
<u>CHAPTER 8</u>	159
<u>CLOSURE</u>	
8.1 Introduction	159
8.2 The experimental study of film cooling	159
8.2.1 Major conclusions	159
8.2.2 Future work on leading edge film cooling	162
8.3 The numerical modelling of film cooling	163
8.3.1 The present contribution	163
8.3.2 Recommendations for future work	167
 NOMENCLATURE	 170
 REFERENCES	 175
 <u>APPENDIX 1</u>	 192
 <u>POTENTIAL FLOW ANALYSIS FOR CYLINDER IN WINDTUNNEL</u>	
 <u>APPENDIX II</u>	 198
 ASME PAPER 80 - WA/HT-26	
 FIGURE CAPTIONS	 207
 FIGURES	 215

CHAPTER 1

INTRODUCTION

1.1 Preliminary remarks

The work presented in this thesis concerns the interaction between jets emerging from orifices in a wall with a crossflowing mainstream. This flow arises in a number of situations which are of practical interest including vertical take-off aircraft gas turbine combustors and the film cooling of the rotor and stator blades of gas turbine engines. In the transition from vertical take-off to forward flight it has been found that the flow from the lift jets of vertical take-off aircraft interferes with the flow around the wing and fuselage leading to a loss of lift. In gas turbine combustion chambers, the hot gases of the primary zone are mixed with jets of cold fluid entering through holes in the combustion chamber wall. Since it is desirable to minimise the overall length of the combustion chamber the mixing between the jets and the hot mainstream flow must be both rapid and thorough to ensure the uniformity of the temperature distribution at the combustion chamber exit. It is the final example however, the film cooling of gas turbine rotor and stator blades, which has motivated the present study.

Film cooling has for many years been employed in cooling the rotor blades and stators of gas turbine engines. Air bled from the compressor is injected through slots or rows of holes in the surface of components exposed to high gas temperatures. Such a method of cooling allows operation of engines at high temperatures, improving the efficiency of the engine cycle, however, there are drawbacks. Although cooling by air bled from the compressor is the only currently practicable method for

aircraft gas turbines it involves an inevitable reduction in the actual efficiency gain. Further losses arise in the case of film cooling due to the mixing of the injected flow with that over the surface of the component. Film cooling can also give rise to large thermal gradients in the material of the cooled component leading to large stress concentrations with the possibility of cracking the component. Clearly it is important to minimise the air bled from the compressor for cooling purposes, whilst ensuring that the film cooling maximises thermal protection with minimal temperature gradients, and aerodynamic loss. Further discussion of the design of blade cooling systems for gas turbines can be found in Halls (1971), Fullagar (1973), and Elovic and Koffel (1979).

A typical film cooled turbine rotor blade is shown in figure 1.1. The film cooling air is ejected from several points around the component from the leading edge, where the surface curvature is large and the boundary layer on the surface of the component is laminar and thin, to the trailing edge, where surface curvature is small and the boundary layer is turbulent or transitional and relatively thick. The Mach number distribution around the component, also shown in figure 1.1, indicates that the flow is transonic and that the flow experiences severe accelerations which, locally, may exceed the levels normally associated with relaminarisation of turbulent boundary layers. The flow around the component is highly disturbed, either from combustion chamber generated turbulence, or from the influence of the wakes of upstream blades.

A more complete description of the parameters influencing the film cooling performance of gas turbine blades is given in Table 1.1.

PARAMETER	RANGE
V_s/V_∞	0.2 to 2.0
ρ_s/ρ_∞	1.5 to 2.2
$\rho_s V_s/\rho_\infty V_\infty$	0.3 to 4.0
Re_s	5×10^3 to 5×10^5
Re_{τ_c}	1×10^6 to 4×10^6
Ra	0 to 5×10^5
M_{∞}	~ 0 to 1.3
$(V_\infty^2/\gamma) dV_\infty/dx$	-5×10^{-7} to 5×10^5
Tu_∞	Up to 20 + %
Boundary Layer State	Laminar, Transitional, Turbulent
δ^*/D	~ 0 to 0.5
R/D	- 100 to 6.0
P/D	4 to 5
t/D	1.5 to 6.0
D	3×10^{-4} m to 8×10^{-4} m
α	30° to 100°

Table 1.1 Ranges of parameters influencing gas turbine blade film cooling.

In spite of the number of parameters involved much progress has been made in establishing their influence by laboratory experiments. This work is discussed in section 1.3 of this chapter. It is clear, however, that attempts to provide empirical correlations representing film cooling performance are not likely to give an adequate description of film cooling for the purposes of turbine blade design. In view of this, attention has focused on the theoretical modelling of film cooling, and a discussion of this work is given in section 1.4.

1.2 Objectives of the present study

The specific objectives of the present work are :-

1. To survey published information relating to film cooling from slots, jets in crossflow and rows of holes with a view to :-
 - i) Provide information of relevance to the film cooling of gas turbine blades.
 - ii) Identify aspects of gas turbine film cooling not adequately represented by published information.
 - iii) Provide data for test cases, against which to assess the numerical methods resulting from the present work.
2. Carry out experimental work to supplement the published data.
3. Develop and assess a numerical method for the prediction of the aerodynamic and heat transfer aspects of :-

- i) Film cooling from a two dimensional slot.
- ii) Jets in crossflow.
- iii) A configuration of interest to gas turbine blade film cooling.

Few studies have investigated the application of film cooling at the leading edge of a gas turbine blade. The heat transfer performance of leading edge films has been studied by Rastogi and Whitelaw (1972), Hanus and L'Ecuyer (1976) and Luckey and L'Ecuyer (1981). However, apart from the flow visualisation study of Russell (1979), the aerodynamics of leading edge films has not been studied. In view of this, leading edge film cooling has been chosen as the subject of the experimental work presented in chapters 2 and 3.

1.3 Discussion of film cooling data

The majority of investigations into the heat transfer performance of film cooling represent the cooling (or heating) effect in terms of a non-dimensional wall temperature defined as :-

$$\eta_{aw} = \frac{T_{\infty} - T_{aw}}{T_{\infty} - T_f} \quad (1-1)$$

and called the adiabatic wall effectiveness.

In some investigations use has been made of the analogy between heat and mass transfer. The results of these investigations are presented in terms of an impervious wall effectiveness which is defined in a similar way, and which is equivalent to η_{aw} .

When employed for gas turbine blade cooling, film cooling is, in general, used in conjunction with other forms of cooling, so that the wall temperature is not represented by T_{aw} .

In order to determine the wall heat flux the following definition of heat transfer coefficient is commonly employed :

$$h_f = \frac{\dot{q}_w''}{T_{aw} - T_w} \quad (1-2)$$

This heat transfer coefficient will generally not be the same as the heat transfer coefficient without the film (h_o). Some investigators present measurements of h_f/h_o in addition to η_{aw} , to allow the use of the data on non-adiabatic walls. This data must be used with care, however, since h_o is dependent on the state of the boundary layer at the point in question so that the experimental data must be representative of the configuration of interest.

A minority of investigators present their results in terms of an isothermal wall effectiveness

$$\eta_{is} = 1 - \frac{\dot{q}_f''}{\dot{q}_o''} \quad (1-3)$$

This is equivalent to η_{aw} only if $h_f/h_o = 1$ and $T_s = T_w$ so that it is necessary to make a clear distinction between the two definitions.

1.3.1 Two dimensional film cooling

The flowfield set up by the discharge from an angled slot is shown schematically in figure 1.2. The fluid injected through the slot presents a blockage to the mainstream flow, causing regions of high pressure upstream of the slot and low pressure downstream. At sufficiently large angles of injection (α) and injection rates, the flow separates at the downstream corner of the slot to form a recirculation bubble.

The flow reattaches to the surface at some point downstream of the slot and eventually returns to boundary layer form. Wygnanski and Newman (1968) find that the length of the recirculation bubble downstream of the slot is characterised by the ratio of the momentum fluxes of the jet and mainstream flows ($\rho_j V_j^2 / \rho_\infty V_\infty^2$). In spite of this however, the majority of investigators characterise the heat transfer performance of the slot in terms of the mass flux ratio M ($\rho_j V_j / \rho_\infty V_\infty$). This may be adequate far downstream of the slot, but it cannot characterise the near-slot region which is dominated by the recirculation. The results of Pedersen (1972) suggest that film cooling flows are in fact characterised by ρ_j / ρ_∞ and V_j / V_∞ ($=J$) individually, rather than in combination and this form of presentation has been employed in the present work.

1.3.1.1 Adiabatic wall effectiveness (η_{aw})

The adiabatic wall temperature distribution downstream of a slot may be described in terms of the temperature of the fluid in the vicinity of the wall. Immediately downstream of the slot, the near-wall fluid has been convected upstream within the recirculation. As a result of mixing between the jet and mainstream flows this fluid is not of unity effectiveness. Thus progressing downstream from the slot, there is a rapid fall from the unity effectiveness fluid at the slot exit to the effectiveness level of the fluid in the recirculation, which is largely dependent on the downstream extent of the recirculation (x_R / S). Further downstream, mixing between the injected flow and the mainstream results in a steady decrease in η , which asymptotes to zero as the downstream distance x tends to infinity.

Sivasegaram (1967) has measured η_{aw} downstream of several slot configurations with angles of injection ranging from 0 to 90 degrees. The results indicated that there was an optimum velocity ratio $\mathcal{T} = 1.0$, at which η_{aw} was maximised regardless of the angle of injection.

1.3.1.2 Heat transfer coefficient (h_f)

The injection of film cooling air influences the heat transfer coefficient defined by (1-2) through two opposing effects : The mixing between the injected flow and the mainstream creates turbulence which tends to enhance h_f ; the thickening of the wall boundary layer by the injected flow, which tends to decrease h_f . At low injection rates and angles of injection, α , the two effects are of similar magnitude and little alteration of h_f may be observed, Hartnett, Birkebak and Eckert (1961a). At higher injection rates ($\mathcal{T} > 0.2$) and angle of injection ($\alpha > 20^\circ$) significant enhancement in the heat transfer coefficient in the vicinity of the slot may be observed, Hartnett Birkebak and Eckert (1961b), and Foster (1973). As the injection induced turbulence decays downstream of the slot the heat transfer coefficient enhancement decays also. Scesa (1954) found no significant difference between the heat transfer coefficients with and without injection beyond $X/S = 50$ downstream of a normal injection slot. The measurements of Foster (1973) indicated that the heat transfer coefficient enhancement increases with increasing $\mathcal{T}$.

1.3.1.3 Density ratio ($\rho_{\mathcal{T}} / \rho_{\infty}$)

The influence of density ratio on η_{aw} has been studied by Matthews (1972). Comparisons of results for $\rho_{\mathcal{T}} / \rho_{\infty} = 1$ and 2 at the same velocity ratio showed no consistent trend however, comparisons made at fixed massflux ratio ($\rho_{\mathcal{T}} V_{\mathcal{T}} / \rho_{\infty} V_{\infty}$)

indicated that η_{aw} increased with the density ratio. This arose because for a fixed massflux, the increased density of the injected flow reduced its momentum and with this the penetration of the jet and the extent of the recirculation.

1.3.1.4 Injection geometry

Wyganski and Newman (1968) and Matthews (1972) studied the influence of angle of injection (α) on the reattachment length (x_R) downstream of the slot. They found that x_R increased rapidly with increasing α . Indeed for a 20 degree slot Metzger, Carper and Warren (1972) found no evidence of recirculation for $J < 1$. The influence of α on η_{aw} has been studied by Sivasegaram (1967), Artt (1970) and Matthews (1972). They found that the rapid reduction of η_{aw} immediately downstream of the slot increases with α and that the overall level of η_{aw} is reduced. This is consistent with the observed trends in the reattachment length. Metzger and Fletcher (1969) showed greater enhancement of heat transfer coefficients with injection at $\alpha = 60$ degrees compared with $\alpha = 20$ degrees.

Minty (1973) investigated the influence of slot aspect ratio (Cross-stream width/ S) on the effectiveness, covering a range of aspect ratios from 1 to 48, and found that η_{aw} increased with increasing aspect ratio. Clearly for low aspect ratios the flow becomes three-dimensional, however Artt (1970) found significant three-dimensionality in the flow from a slot of aspect ratio 48. Consequently most investigators ensure that the slot spans the full width of the tunnel, however, because of the side wall boundary layer, even this may be inadequate and additional side plates are fitted to prevent three-dimensionality, Wyganski and Newman (1968) and Foster (1973).

Weighardt (1946), Hartnett et al (1961a, 1961b) and Mayle et al (1977) employed a slot geometry which differed from other investigators in that the downstream edge of the slot was blended to the downstream surface by a smooth curve. This will suppress the recirculation and change the character of the downstream flow.

1.3.1.5 Upstream boundary layer (δ^*)

The influence of the upstream boundary layer thickness on the film effectiveness of angled slots has been studied by Matthews (1972). The observed effects are small and, since they appear to be inconsistent, may have been influenced by experimental error.

1.3.1.6 Mainstream turbulence

The influence of mainstream turbulence intensity on slot film cooling effectiveness has been studied by many workers, however, the observed effects are in conflict. For example Carlson and Talmor (1968), Burggraf and Huffmeier (1973), and Marek and Tacina (1975) observed a decrease in η_{is} or η_{aw} with increasing mainstream turbulence intensity, but Minty (1973) observed no effect on increasing Tu_{∞} .

Sturgess and Lenertz (1977) have described this behaviour in terms of the integral scale of the turbulence (Λ).

Where the integral scale of the mainstream turbulence Λ_{∞} is greater than that generated by the film ejection Λ_f , then η_{aw} is degraded. Where Λ_{∞} is less than Λ_f then η_{aw} is unchanged.

1.3.1.7 Pressure gradient $\left\{ k = \frac{1}{V_\infty^2} \frac{dV_\infty}{dx} \right\}$

Wyganski and Newman (1968) and Foster (1973) examined the influence of adverse pressure gradient on the recirculation downstream of angled slots. They found that the recirculation was increased in size relative to the zero pressure gradient condition. In view of the arguments of section 1.3.1.1 this should tend to decrease η_{aw} , however, Foster (1973) found no appreciable change in either η_{aw} or heat transfer coefficients. Hartnett, Birkebak and Eckert (1961b) determine the influence of favourable pressure gradients of $k = 1 \times 10^{-6}$ and 2×10^{-6} on η_{aw} and heat transfer coefficient downstream of a non-separating slot configuration. They find that the heat transfer coefficient enhancement caused by the film injection was little effected, but that η_{aw} was decreased by the favourable pressure gradient. If this trend is applicable to adverse pressure gradients, the negative result of Foster (1973) may be understood in terms of a balance between the decrease in η_{aw} arising from the increase in the size of the recirculation and an increase in η_{aw} arising from the adverse pressure gradient.

Warren and Metzger (1972) investigated the effect of injection into a laminarising mainstream flow ($k = 5 \times 10^{-6}$). They observed that the injection provoked transition to turbulent flow and resulted in large increases in the heat transfer coefficients downstream of the slot.

1.3.1.8 Surface curvature

Louis et al (1974) studied the isothermal wall effectiveness of a flat surface and a concave surface downstream of a 20° slot. They found that η_{is} was decreased on the concave surface

relative to the flat surface. This result was confirmed by Mayle et al (1977) who also found that, relative to a flat surface η_{aw} was increased on a convex surface. These results are a consequence of the action of curvature on turbulent mixing, Bradshaw (1973), and Görtler vortex activity on the concave surface.

1.3.1.9 Compressibility

Papell (1960) has obtained measurements of η_{aw} for a 90° slot at mainstream Mach numbers of 0.2, 0.5 and 0.7. A slight increase in η_{aw} with Mach number may be observed in his results.

1.3.2 Three dimensional film cooling

The flowfield set up by a jet in crossflow is shown schematically in Figure 1.3. Upstream of the jet the mainstream is retarded and divides to pass on either side of the jet which is bent by the mainstream by an amount dependent on the relative momenta of the jet and mainstream flows. Downstream, the flow is dominated by a pair of contra-rotating streamwise vortices which, where the jet lifts off the surface, are enhanced by entrainment into the underside of the jet, and which may persist for a considerable distance downstream. Where the jet is ejected from one of a row of holes, lateral spreading causes the jet flow from adjacent holes to merge at a point downstream, dependent on the hole spacing (P/D). Thereafter, turbulent mixing combines with the action of the streamwise vortex to cause lateral mixing of the jets so that the temperature and streamwise velocity profiles became two-dimensional far downstream of the injection point.

1.3.2.1 Adiabatic wall effectiveness

The adiabatic wall temperature distributions set up by ejection from single holes and rows of holes have been studied extensively. The measurements of Goldstein, Eckert and Ramsey (1968) and Eriksen (1971) are typical. At low injection rates the flow from the hole remains in contact with the wall and the centreline ($Z/D = 0.0$) effectiveness (η_{aw}) decreases monotonically with downstream distance as the injected fluid mixes with the mainstream. At off centreline locations, $Z/D \geq 1.0$,

η_{aw} is increased in the vicinity of the hole, due to lateral spreading of the jet, but decreases further downstream as the jet mixes out. At high injection rates the jet lifts off the surface, and the mainstream fluid penetrates beneath it, so that there is little wall cooling near the ejection point. Provided α and J are not too large, the jet reattaches to the surface at some point downstream and the centreline effectiveness rises.

For single holes, Goldstein, Eckert and Ramsey (1968) found that η_{aw} was maximised at $J = 0.5$, regardless of the angle of injection. The results of Eriksen (1971) for a row of holes were similar to those of Goldstein et al, except that downstream of the point at which the jets merge, η_{aw} increases monotonically with J .

1.3.2.2 Heat transfer coefficient

The heat transfer coefficient in the vicinity of a film cooling hole is modified by the injection as a result of two opposing effects : The turbulence arising from the mixing between the jet and the mainstream which tends to increase h_f ; and the thickening of the wall boundary layer, resulting from the addition of the injected flow, which tends to reduce h_f .

At low angles of injection (α) and velocity ratios (J) the two effects balance and h_f is little different from the heat transfer coefficient without injection. The measurements of Eriksen (1971) indicated no enhancement of heat transfer for $\alpha = 35$ degrees and $J < 1.0$. As J is increased the enhancement due to the mixing generated turbulence becomes dominant and h_f increases. Eriksen (1971) indicated enhancement of h_f in the vicinity of the holes of up to 35% for $J = 2.3$ and $\alpha = 35$ degrees.

1.3.2.3 Density ratio (ρ_r / ρ_∞)

Foster and Lampard (1974) and Pedersen (1972) have investigated the influence of ρ_r / ρ_∞ on η_{aw} . Foster and Lampard (1974) found that increasing ρ_r / ρ_∞ from 1 to 4.2 increased the effectiveness of a row of 90° holes ejecting at $J = 0.7$ downstream of $x/D = 20$. At a fixed mass flux ratio however, increased density ratio caused η_{aw} to increase close to the holes. Further downstream the enhancement decreased so that at $x/D \sim 100$ η_{aw} was virtually independent of density ratio.

For a fixed mass flux the increased ρ_r / ρ_∞ decreases the momentum of the injected flow reducing the initial penetration and mixing.

The extensive work of Pedersen (1972) on a row of holes at 35 degrees covers a range of density ratios : $0.14 \leq \rho_r / \rho_\infty \leq 4.2$ and mass flux ratios $0.2 \leq M \leq 2.0$. In addition to the trends observed by Foster and Lampard (1974) the results showed that, for a given density ratio, the effectiveness was maximised at $J \approx 0.45$ over the full range of density ratios studied. The merging of the jets downstream of the injection point also appeared to be controlled by the velocity ratio and occurred at $J \approx 1.5$ over a wide range of density ratios. These results

suggest that the parameters characterising the flow are actually the density ratio ρ_r/ρ_∞ and velocity ratio J rather than the traditionally accepted mass flux ratio M . Pedersen's results showed that the effectiveness was characterised by M only at the low injection rates where the jet remained attached to the surface.

1.3.2.4 Injection geometry

The influence of the angle of injection (α) on the flow from a row of film cooling holes has been studied by Colladay and Russell (1975) using helium bubble flow visualisation. For injection at 90 degrees the jet was observed to separate from the surface even at the lowest injection rate employed, $J = 0.3$. The bubble paths were tortuous and exhibited sharp kinks indicating high turbulence intensities. There was some evidence of flow reversal. For injection at 30° the streaklines indicated much reduced turbulent mixing and spreading of the jet, and the vortex structure was not so evident as for the 90 degree holes. The film separated from the surface for $J \geq 0.8$.

The temperature profiles downstream of single holes ejecting at 35 and 90 degrees to the surface have been measured by Ramsey and Goldstein (1971). These show that, for mass flux ratios, M , of 1.0 and 2.0, the jet from the 90° hole penetrates 60% further from the surface, has a lower peak effectiveness, and the adiabatic wall effectiveness on the centreline, $Z/D=0.0$, is up to 40% lower than that of the 35° hole. The measurements of Goldstein, Eckert and Ramsey (1968) showed however that, because of the increased lateral spread of the jet from the 90° hole the adiabatic wall effectiveness at off centreline locations was increased relative to the 35° hole.

For rows of holes Goldstein, Eriksen and Ramsey (1978) and Foster and Lampard (1979) observed that, in terms of the laterally averaged effectiveness (η_{aw}), low angles of injection were best at low injection rates, but that at high injection rates large angles of injection gave more cooling. Unfortunately Foster and Lampard (1979) found that the increased mixing associated with the larger angles of injection caused greater aerodynamic loss and Eriksen (1971), and Metzger and Fletcher (1969) found greater enhancement of heat transfer coefficients.

Colladay and Russell (1975) also reported flow visualisation on a row of holes in which the jet was directed laterally across the flow at an angle (γ). For the configuration tested : $\alpha = 30^\circ$, $\gamma = 45^\circ$ the flow downstream of each jet was characterised by a single vortex, which formed at $J = 0.3$ and became most pronounced for $0.7 < J < 0.9$. The strong vortex motion kept the injected fluid attached to the wall and very little increase in penetration was observed on increasing J from 0.3 to 0.9. Goldstein Et al (1969) found that holes inclined in the lateral direction gave increased peak value and lateral spread of η_{aw} for $M > 0.5$.

Sterland and Hollingsworth (1975), studied the influence of hole pitching on the penetration of a row of jets in crossflow. They found that the penetration was minimised for a pitch-diameter ratio of 4. Liess and Carnel (1971) and Foster and Lampard (1979) investigated the influence of hole pitching on η_{aw} . They found that reduced P/D gave improved lateral coverage at all mass flux ratios and suppressed lift off at higher injection rates.

Double row film cooling geometries have been studied by Jabbari (1973), Kruse (1974), Louis et al (1974) and Smith Jones and Schultz (1974). In most cases a staggered hole arrangement has been used in which the downstream holes were located mid-way between the holes of the upstream row. This configuration was found to give improvement in η_{aw} in excess of the increased mass flow requirement of the double row. Bergeles (1976) observed that this occurred because of the interference to the vortex structure caused by the injection from the downstream row which reduced the mixing with the mainstream flow. Jabbari (1973) observed that injection from a staggered double row configuration was significantly superior to an in-line geometry.

Goldstein, Eckert and Burggraf (1974) investigated the effect of shaping the film cooling holes by widening the hole exit. This gave reduced jet penetration, and higher peak and laterally averaged effectiveness which were caused partly by the widening of the hole and partly because the increased exit area of the hole reduced the injection velocity and tends to suppress lift off.

1.3.2.5 Upstream boundary layer

The wall boundary layer of the mainstream flow approaching a row of film cooling holes on a gas turbine blade may be either laminar or turbulent and of varying thickness dependent on the location of the injection on the component. Liess (1974) compared the effect of injecting films into both laminar and turbulent boundary layers and observed no significant differences in η_{aw} . The heat transfer coefficient measurements of Lander, Fish and Suo (1972) showed that the injection of film cooling air into a laminar boundary layer provoked the transition of the boundary

layer to turbulent flow causing large increases in the heat transfer coefficients.

Many workers have studied the influence of the thickness of the upstream boundary layer on film cooling effectiveness . Although the results show the same general trend of reduced effectiveness with increase in boundary layer thickness, there are inconsistencies in the magnitude of the effect. Pedersen (1972) explained these results by relating the jet penetration and the boundary layer thickness: At low and high injection rates the jet is either flattened instantly against the surface, or passes rapidly through the boundary layer. In either case the boundary layer thickness has little effect. At intermediate injection rates however, where the jet penetration and the boundary layer thickness are of the same order, the film effectiveness may be significantly affected. The results of Liess (1974) support this interpretation. Kadotani and Goldstein (1979) observed that increasing the boundary layer thickness increased the strength of the streamwise vortices so that although significant changes occurred in the centreline effectiveness, the change in the lateral average value is small.

Eriksen (1971) observed a slight decrease in h_f with increased boundary layer thickness.

1.3.2.6 Mainstream turbulence

The influence of the intensity and length scale of mainstream turbulence on the interaction between the jets and mainstream flow and their film cooling performance, has been investigated by Kadotani and Goldstein (1977a, 1977b). They found that, at small length scales ($L/D \sim 0.1$), mainstream turbulence had little effect on the flow, but at larger length

scales ($L/D \sim 0.3$) the film cooling jets mixed with the mainstream and the streamwise vortices associated with the injection were suppressed. The influence of mainstream turbulence on the film cooling performance is dependent on the injection rate, M . For $M \leq 0.2$ an increase in turbulence intensity and length scale decreased the laterally averaged effectiveness η_{aw} through enhanced mixing. For $M > 1.0$ the laterally averaged effectiveness, η_{aw} , increased with increasing intensity due to the reduction of the jet penetration, but decreased with an increase in length scale. At intermediate injection rates the lateral distribution of η_{aw} was significantly affected by the length scale of turbulence because of the modification of the streamwise vortices.

1.3.2.7 Pressure gradient

The measurements of Foster and Lampard (1974) and Jabbari (1973) showed that favourable mainstream pressure gradients tended to reduce the effectiveness far downstream of the holes. Liess found also, that the rate of decrease of η_{aw} with downstream distance is reduced in the region of accelerating flow. Liess ascribed this behaviour to the influence of the acceleration on the streamwise vortices, however, the suppression of the turbulent mixing will also be important. Jabbari (1973) found little influence of accelerations of up to $k = 1 \times 10^{-6}$ on heat transfer coefficients for double row configurations. The results of Liess (1974) show however, that the heat transfer coefficients were suppressed by accelerations of up to $k = 5 \times 10^{-6}$, but enhanced downstream of the region of accelerating flow. Aided by flow visualisations Liess argued that the streamwise vortices were suppressed by the acceleration, reducing h_f , but that

downstream of the acceleration the vortices developed suddenly, increasing h_f .

The measurements of Launder and York (1974) showed an increase in η_{aw} with favourable pressure gradient. This occurred because the film cooling jets were laminar and the acceleration delayed the transition to turbulent flow so that the mixing with the mainstream flow was reduced.

1.3.2.8 Surface curvature

Nicolas and LeMeur (1974) and Ito, Goldstein and Eckert (1978) have studied the influence of curvature on the flow from film cooling holes by considering the radial equilibrium of the coolant flow. Nicolas and LeMeur (1974) predict that, dependent on the film to mainstream momentum flux ratio and the sign of the wall curvature, the film effectiveness may be increased or decreased. Measurements of slot injection on a concave wall supported this analysis but those on a convex wall did not. Ito et al (1978) extended this approach to include the effect of the angle of injection. Measurements on a turbine blade for injection at unity density ratio were in good agreement with the analysis, however, for $P_s/\rho_\infty = 2$ the analysis broke down near the holes on the concave surface. The influence of curvature on the turbulent mixing of film cooling from rows of holes has not been addressed.

1.3.2.9 Compressibility

Liess (1974) and Smith, Jones and Schultz (1974) studied the influence of the mainstream flow Mach number on η_{aw} and found that, provided $M_\infty < 0.7$, no effect was observed.

1.3.2.10 Blade environment

Lander, Fish and Suo (1972) have obtained measurements of the adiabatic wall effectiveness and heat transfer downstream of double row film cooling configurations on turbine vane aerofoils in cascade. The heat transfer results showed large increases in heat transfer coefficients with film injection caused by the transition of the vane surface boundary layer to turbulent flow. Gauntner (1977) presented measurements of the adiabatic wall effectiveness of two five row film cooling hole arrays on the suction surface of a gas turbine nozzle guide vane in cascade. An array of laterally inclined film cooling holes was found to be less sensitive to pressure gradient, did not lift off the surface and gave improved cooling performance compared with an in-line array of holes.

In many cases film cooling air is ejected from several points around the periphery of the component so that it is necessary to determine the influence on the performance of a film cooling row of the injection from upstream rows.

Afejuku, Hay and Lampard (1980) have studied the additive nature of two rows of film cooling for both in-line and staggered hole arrays with various row spacings. They found that the simple superposition hypothesis of Sellers (1963) could be used with safety for designs using staggered holes but that this was inadequate for in-line configurations with a row spacing of $10 D$ or less. Dring, Blair and Joslyn (1978) studied film cooling on a rotating cascade of blades. Their flow visualisations showed that, as a result of the low velocities on the pressure surface, the film was centrifuged outwards. The cooling performance of suction surface films was found to be

in agreement with the static cascade measurements of Ito et al (1978) but pressure surface films were found to have inferior performance with rotation. This was attributed to a combination of the effects of surface curvature and the radial flow. A more likely cause, however, is the effect of the Coriolis force arising from the radial motion of the film, which will tend to move it away from the surface.

Hartsel (1970) presented measurements of the aerodynamic loss, and distributions of static pressure, adiabatic wall effectiveness and heat transfer coefficients for a turbine aerofoil with five rows of film cooling holes on each surface. The greatest aerodynamic losses were found for ejection from holes normal to the surface and, although lateral inclination reduced this loss, streamwise inclination was found to give by far the greatest loss reduction. In a later publication, Hartsel (1972), a simple model based on constant pressure mixing was proposed, which allowed the estimation of the aerodynamic loss caused by film cooling.

1.3.3 Film cooling at a leading edge

No quantitative data describing the aerodynamics of leading edge film cooling has been published to date, and the available information is of a qualitative nature only. Russell (1979) presented smoke flow visualisations of the flow from a row of holes in the surface of a cylinder in crossflow. In spite of inclination of the holes along the span of the cylinder, little spanwise deflection or spreading of the jets was observed. For $M > 0.4$ the jets remained approximately constant in width as they flowed around the cylinder, but for $M < 0.4$ the jets necked 1 diameter downstream of the hole. Spanwise inclination

of the holes at 45 degrees gave greater penetration than holes at 30 degrees. Increasing the angle between the nominal stagnation point and the point of injection increased the tendency of the jets to lift off the surface. Sasaki et al (1976) presented flow visualisations of four row leading edge cooling configurations. They found that at low injection rates the films remained attached to the surface but that for $M > 1$ the film lifted off the surface. Spanwise inclination was found to suppress lift off and gave better coverage of the surface. Measurements of the adiabatic wall effectiveness confirmed the superior performance of the spanwise inclined film cooling holes.

Rastogi and Whitelaw (1972) presented measurements of η_{aw} for film cooling from a single row of holes normal to the surface of a cylinder in cross-flow. Subsequently Taylor and Whitelaw (1975) have extended this work to include rows of holes inclined spanwise at 25 degrees to the surface. They found that the spanwise inclined holes had an improved cooling performance relative to the normal holes, except for injection on the stagnation line at $J > 0.6$ and that increased density ratio increased the cooling effect for a given velocity ratio. Results for injection at $\Theta_s = 0, 20, \text{ and } 45$ degrees from the nominal stagnation point showed that, at a given injected flow rate, η_{aw} increased with Θ_s . Taylor and Whitelaw also found that fitting a splitter plate to the rear of the cylinder had no effect on η_{aw} .

Luckey et al (1977) measured the isothermal wall effectiveness of a single row of spanwise inclined holes on a cylinder in crossflow with ejection at $\Theta_s = 20, 30 \text{ and } 40$ degrees. The results showed the existence of significant spanwise variations

in effectiveness which, curiously, were largest for the lowest angle of injection.

The laterally averaged effectiveness was found to reduce at a fixed local massflux ratio as Θ_s was decreased. This was attributed to the acceleration of the flow around the surface. Hanus and L'Ecuyer (1976) present isothermal wall effectiveness results for ejection at the leading edge of a 3 x size model turbine vane. The results for holes inclined in the spanwise direction at 18 and 35 degrees to the surface were found to be almost identical up to the point at which the jets lift off the surface. For a fixed local massflux ratio the cooling effect was found to decrease on moving the ejection point nearer to the stagnation point. Pedersen (1972) observed an optimum velocity ratio of 0.4 at which the effectiveness of film cooling of a flat plate by a streamwise inclined row of holes was maximised. Hanus and L'Ecuyer (1976) also observe that the cooling effect is maximised at a local velocity ratio of 0.4 and suggest that this is of some, as yet unknown, significance in determining the conditions for optimum film cooling effectiveness from rows of holes.

1.4 Theoretical models of the film cooling process

1.4.1 Two-dimensional models

One of the earliest theoretical models of film cooling was that of Tribus and Klein (1953) in which the coolant flow was represented by a line source of heat at the ejection point, the strength of which equalled the enthalpy flux of the injected flow. The adiabatic wall temperature downstream of the heat source was determined using the kernel developed by Rubesin (1951) to predict the heat transfer and temperature distributions along

non-isothermal walls with turbulent boundary layers. The model ignores the effect of the mass injection on the mainstream flow and breaks down as $x \rightarrow 0$ where $\eta \rightarrow \infty$.

Kutateladze and Leont'ev (1963), Librizzi and Cresci (1964), and Stollery and El-Ehwany (1965) proposed models of film cooling based on the following sweeping assumptions :-

- i) Downstream of the injection point, the entrainment of mainstream fluid into the mixing layer may be approximated by the growth of a turbulent boundary layer represented by a $1/7$ th power law profile.
- ii) The injected flow mixes with the mainstream to produce a uniform temperature across the mixing layer which approximates to the adiabatic wall temperature.

Spalding (1965), Mayle and Kopper (1976), and Goldstein and Haji-Sheikh (1967) propose modifications to these and similar models in order to improve the agreement with measurement, however, even in the most developed form, the models are unable to predict accurately the adiabatic wall effectiveness within $x/s \sim 50$ of the ejection point.

For tangential slot geometries it is possible to employ numerical methods which solve the equations of motion in boundary layer form. For example Kacker Pai and Whitelaw (1969) employ a modified version of the Spalding Patankar (1967) scheme to calculate the flow development downstream of tangential slots with considerable success. For slots ejecting at an angle to the surface, however, the presence of the flow reversal necessitates the solution of the elliptic equations governing fluid flow. Matthews (1972) has presented solutions of these equations for

angled slots at 30° and 45° to the surface. The numerical method employed was based on the streamfunction-vorticity formulation of the governing equations, Gosman et al (1969). Reynolds stresses were represented by the $k-\epsilon$ turbulence model of Launder and Spalding (1972). Two features of the calculations are undesirable :

- 1) Although the solution method was capable of computing the development of the flow upstream of the slot, Matthews employed experimental measurements to specify the upstream boundary condition immediately upstream of the slot.
- 2) In order to overcome the difficulty of specifying the boundary value of vorticity at a corner Matthews assumed that the separating streamline at the downstream edge of the slot continues into the flow without deviation. In fact the streamline will bend and the assumption of non-deviation could have a significant influence on the predicted results.

Matthews (1972) obtained good agreement with experiment although the method tended to underpredict the velocity peak downstream of the slot.

1.4.2 Three-dimensional models

One of the simplest models to describe the adiabatic wall effectiveness downstream of a film cooling hole was the modification of the heat source model of Tribus and Klein (1953) due to Eriksen, Eckert and Goldstein (1971). The jet was replaced by a point heat source at some height above the surface in a uniform velocity stream and an image source the same distance below the wall gave the adiabatic wall condition.

The height of the source and a turbulent thermal diffusivity have to be specified empirically. The model broke down for $M > 0.5$. Further developments of the heat source model have been described by Kadotani and Goldstein (1979), however, since no attempt is made to represent the aerodynamics of the flow it is unlikely to have general application.

A large number of investigators have studied jets in cross-flow in connection with VSTOL aircraft and atmospheric plumes. Several workers have proposed potential-flow or semi-empirical methods in which the equations of motion were integrated along the path of the jet, for example : Abramovich (1963), Keffer and Baines (1963) Bowley and Sucec (1969), and Campbell and Schetz (1973). Further details of these methods may be found in NASA (1969). More recent developments of these models have included : Hwang and Fletcher (1978), and Adler and Barron (1979). Baker and Manhardt (1979) described a method in which a viscous finite element scheme, used to calculate the development of the jet, was embedded within an inviscid solution for the mainstream flow around the jet. A solution of the full flowfield was obtained by iteration between the jet and mainstream flow solutions. The viscous solution employed an algebraic stress turbulence model based on the work of Launder, Reece and Rodi (1975) and Launder and Spalding (1972).

Whilst the methods described above are adequate for $J > 3$ or 4, the neglect of the influence of the wall causes them to break down at the lower injection rates appropriate to film cooling. Herring (1974), Crawford, Kays and Moffat (1980), and Wang, Papell and Graham (1981) have attempted to model film cooling flows by combining a semi-empirical treatment of the

jet flow with a two dimensional solution of the wall boundary layer. Crawford, Kays and Moffat (1980) obtained good agreement between their model and spanwise averaged measurements of full coverage film cooling but, like Wang, Papell and Graham, found that a single prescription of the empirical constants could not represent a range of film conditions.

None of the three dimensional models described to this point are capable of defining the details of the flowfield set up by jets in crossflow. For example : Crawford, Kays and Moffat (1980) made a clear distinction between their model of film cooling, which they termed a simulation, and more complex methods which are capable of describing the full details of the flow, termed a prediction. In order to predict the flow from film cooling holes in this sense it is necessary to cope with the elliptic nature of the equations governing the flow.

Bergeles (1976) employed a semi-elliptic calculation method with a modified version of the $k-\epsilon$ turbulence model of Launder and Spalding (1972) to calculate the flow from a single hole and rows of holes in crossflow. The semi-elliptic scheme was an extension of three-dimensional boundary layer solution techniques which allowed for upstream effects in the pressure field. The method could not cope with flow reversal in the streamwise direction, however, and was therefore restricted to low injection rates ($J \leq 0.1$ for injection normal to the surface). Rodi and Srivatsa (1980) described an extension to the semi-elliptic techniques in which a localised region of the solution was treated in a fully elliptic manner. This allowed the solution method to cope with small regions of flow reversal. Predictions were presented for normal injection with $J \leq 0.3$.

Chien and Schetz (1975) solved numerically the steady three-dimensional Navier Stokes equations written in terms of velocity-vorticity, for a turbulent buoyant jet in crossflow. The calculations were completed on a finite difference grid of $24 \times 22 \times 22$ nodes and employed a constant Prandtl eddy viscosity model. Good agreement with jet trajectory was obtained, but a CPU requirement of several hours on an IBM 370-158 was quoted.

Patankar, Basu and Alpay (1977) and Crabb (1979) solved the Navier-Stokes equations in primitive-variable form using finite differences to predict the jet in crossflow. The $k-\epsilon$ turbulence model was employed in both studies. Jones and McGuirk (1979) applied a similar method to the computation of high velocity jets in a confined crossflow ($5 < J < 10$).

Patankar et al (1977) obtained solutions for the jet in crossflow using a finite difference mesh of $15 \times 15 \times 10$ nodes. Because of the limited number of mesh nodes available the size of the solution^{domain} was minimised by computational experiment. Good agreement was obtained with measured jet trajectory and velocity profile data, however, Crabb (1979) has shown that errors in this work caused by the coarseness of the calculation mesh were masked by the effect of an insufficiently large solution domain.

Crabb (1979) employed a solution mesh of $24 \times 14 \times 14$ nodes to predict his laser doppler anemometry measurements. In this work the solution domain boundaries were determined by the dimensions of the wind tunnel employed so that the solution mesh is relatively coarse. This is reflected in the results obtained which are in poor agreement with the measured data.

1.4.3 Film cooling on curved surfaces

Rastogi and Whitelaw (1970), Dvorak (1973) and Folley and Whitelaw (1976) have presented boundary layer solutions of wall-jet flows over curved surfaces, using modified forms of the Spalding-Patankar (1967) scheme and Prandtl's mixing length turbulence model. The predictions showed consistently that η_{aw} was reduced on concave surfaces and increased on convex in comparison with a flat surface. Up to the separation point good agreement with experiment was obtained.

Irwin and Arnot-Smith (1975) described calculations of wall-jets on curved surfaces in which a Reynolds stress turbulence model based on that of Launder, Reece and Rodi (1975) was employed. Again good agreement with experiment was obtained.

1.5 Outline of the thesis

The next two chapters of the thesis are concerned with the experimental study of leading edge film cooling, the apparatus and measurement techniques being described in chapter 2 and the results obtained are presented in chapter 3. The theoretical basis of the numerical methods employed to predict the film cooling flows is described in chapter 4. The modification and application of these methods to the prediction of the aerodynamic and heat transfer aspects of film cooling from two-dimensional slots, jets in crossflow, and at a leading edge are described in chapters 5, 6 and 7 respectively. Conclusions and recommendations stemming from this work are discussed in chapter 8.

CHAPTER 2

APPARATUS AND MEASUREMENT TECHNIQUES

2.1 Introduction

The experimental part of the present study comprises an investigation of the flow set up by the ejection of air through rows of holes in the surface of a cylinder in crossflow. The experiment is arranged to model some of the major features of a gas turbine rotor blade with leading edge film cooling. Several workers have already investigated similar configurations (c.f. section 1.3.3), however, geometries representative of rotor blades have not been investigated in detail, and the aerodynamics of leading edge film cooling has not previously been studied in a quantitative manner.

The objectives of this work are to provide detailed information on the flow for turbine design and provide data with which to validate numerical predictions. The work includes flow visualisation together with measurements of the velocity profiles and static pressure distributions on the surface of the cylinder and concentrations of a tracer gas (Helium) injected with the secondary flow. The passive scalar results allow the dispersion of the secondary fluid to be quantified and, through analogy between heat and mass transfer, allow the cooling effect of the secondary fluid to be determined.

This chapter describes the apparatus and the measurement techniques employed. The results are presented in Chapter 3.

2.2 Apparatus

2.2.1 Windtunnel

The experiments were conducted in an open circuit windtunnel. Mainstream air for the tunnel was drawn from the laboratory by a centrifugal blower driven at constant speed by an electric motor. An adjustable slide valve on the blower inlet controlled the mainstream flowrate. The air was passed from the blower through a diffuser, settling chamber and contraction sections to the test section which was of 300 mm by 200 mm cross-section.

2.2.2 Test piece

The rotor blade leading edge was modelled by a hollow cylinder of 100 mm outside diameter mounted in the test section as shown in figure 2.1(a). A splitter plate was fitted to the rear of the cylinder to prevent vortex shedding which caused probe vibration.

Three rows of film cooling holes 8 mm in diameter were drilled radially into the cylinder on a pitching of 4 hole diameters, and an angular separation of 40 degrees. Four ejection configurations were studied :-

1. Bare cylinder
2. Single row on stagnation line ($\theta_r = 0^\circ$).
3. Two rows at $\pm 20^\circ$ to either side of the stagnation line ($\theta_r = \pm 20^\circ$).
4. Two rows at $\pm 40^\circ$ to either side of the stagnation line ($\theta_r = \pm 40^\circ$).

These are shown in figure 2.1(b)

Each hole configuration was obtained by selectively blocking rows of holes with sellotape, and rotating the cylinder so that the holes were symmetrically disposed on either side of the stagnation line.

The internal cavity of the cylinder was employed as a plenum chamber for the secondary air and, with a 68 mm internal diameter, was large enough to ensure an even distribution of flow from the film cooling holes. The hole length to diameter ratio was 2, near the current gas turbine rotor blade value which is of order $L/D \sim 3$.

A total of 52 static tapping holes 0.5 mm in diameter were drilled in the surface of the test piece to the arrangement shown in figure 2.2.

2.2.3 Secondary air supply

The secondary air was obtained from the laboratory shop air supply and fed to the test cylinder using the system shown in figure 2.3. Air was fed to both ends of the test cylinder to ensure an even distribution of flow from the ejection holes. With this system the jet velocities in the absence of mainstream flow were the same from each hole to within the repeatability of the measurements.

The secondary air flow rate was set up in the absence of mainstream flow. The relationship between the area mean jet exit velocity and the velocity at the centre of the jet was determined for each injection configuration. The desired mean jet velocity could then be obtained by adjusting the regulator to give the corresponding velocity at the centre of the jet.

The secondary flow rate was maintained constant during the test run by maintaining the resulting control pressure constant. Because the effective pressure at the jet exit is dependent on the mainstream velocity and the location of the ejection point on the cylinder, however, the secondary flowrate set without mainstream flow differs from that with mainstream flow. This was minimised by sizing the restrictor to give the highest possible control pressure at the desired flowrate. All the results quoted have been corrected for this effect, the correction being, typically, less than 1%.

2.2.4 Traverse arrangements

Traverse motion was allowed in the X, Y and Z directions, Figure 2.1. In the X and Z directions the positional accuracy of the traverse gear was ± 0.05 mm, and in the Y direction ± 0.1 mm.

As a result of the need to obtain profiles in the R direction, the traversing required motion of both the X and Y slides. In view of this the angular locations of the traverses were chosen so that the relative motion of the two slides was in a simple ratio.

Since it was not possible to mount the traverse gear directly over the test cylinder because of the secondary air feed pipes, the traverse gear was mounted upstream of the cylinder and an offset probe holder employed. The probe holder was sufficiently rigid to ensure that the positional accuracy of the probe location was not significantly greater than that of the traverse gear.

2.2.5 Measurement probes

Two separate probes were employed : A pitot probe to determine profiles of the tracer gas concentration, and a five-hole probe to determine the profiles of the V_r , V_θ , and V_z velocity components around the cylinder. The pitot probe was manufactured from 1.mm O.D. stainless steel hypodermic tubing and the five-hole probe from 0.8mm O.D. tubing, figure 2.4. The overall width of the five-hole probe was 2.5 mm. Bryer and Pankhurst (1971) studied a wide range of pressure probes to determine their sensitivity, accuracy, and range of application. They considered two designs of 5-hole probe : one with an apex angle of ninety degrees, and the other with an apex angle of seventy degrees. The ninety degree apex angle probe gave greater directional sensitivity, and suffered less from Reynolds number effects so that this has been employed in the present work.

For accurate measurements in shear layers the dimensions of the 5-hole probe must be small in comparison with the scale of the shear layer. The smaller the probe, however, the greater the time required for the probe to respond to a change in the flow conditions, and the effect of Reynolds number on the probe calibration is increased. There is, therefore, a minimum size of probe which is practicable for use in any particular application. The probe design shown in figure 2.4(b) represents the smallest that could be used for the present work. These points are discussed further in section 2.4.3.

Both the sampling probe and the 5-hole probe were manufactured so that the stem of the probe was a tight fit in the probe holder. This allowed the probe to be rotated to any

desired angle to the flow, and yet was sufficiently firm to prevent movement of the probe once it was set up.

The probes were designed so that the measurement head was offset from the stem of the probe to prevent the blockage of the flow that would otherwise have arisen from the juxtaposition of the cylinder wall and the stem of the probe, when measuring near the wall.

2.3 Experimental conditions

The experimental conditions employed are compared with those at the leading edge of a typical gas turbine rotor blade in Table 2.1. Further details are given in Table 2.2 which also indicates the measurements obtained at each condition. Because of bistability of the jet flow the full range of measurements were not attempted for the $J = 1.8$ condition of the single row configuration. The velocity profile measurements were only made at the high jet velocity ratios to minimise the effects of the poor spatial resolution of the 5-hole probe.

<u>Parameter</u>	<u>Typical Gas Turbine Rotor Blade</u>	<u>Experimental Condition</u>
L.E. Radius (R)	2.4 mm	50 mm
Film Cooling Hole Diameter (D)	0.39 mm	8 mm
R/D	6.25	6.25
P/D	4-5	4
Mainstream Approach Velocity (V_∞)	200-300 m/s	13 m/s
Approach Mn	0.25	0.04
Mainstream Turbulence intensity (Tu_∞)	8 - 12%	0.5%
Point at which accl'n. parameter, $k < 2.5 \times 10^{-6}$	$\theta = 50-60^\circ$	65°
Reynolds number based on L.E. Diameter	8.5×10^4	8.5×10^4
Rayleigh No.	5×10^5	0.0
Density Ratio (ρ_r/ρ_∞)	~ 1.6	1.00
Velocity Ratio (J)	0.5 - 1.5	0.5 - 1.8

Table 2.1 Comparison between the experimental conditions and those for a typical gas turbine rotor blade leading edge.

Configuration	Condition	Measurement Obtained					
		Smoke Visualisation	Surface Visualisation	Surface Static Pressure	Velocity Profiles	Wall Concentration	Concentration Profiles
Bare Cylinder	No Blow			✓	✓		
Single Row on Stagnation Line	$J = 0.5$	✓	✓	✓		✓	✓
	$J = 1.0$	✓	✓	✓	✓	✓	✓
	$J = 1.8$	✓	✓	✓			
Two Rows at ± 20 degrees	$J = 0.5$	✓	✓	✓		✓	✓
	$J = 1.0$	✓	✓	✓		✓	✓
	$J = 1.8$	✓	✓	✓	✓	✓	✓
Two Rows at ± 40 degrees	$J = 0.5$	✓	✓	✓		✓	✓
	$J = 1.0$	✓	✓	✓		✓	✓
	$J = 1.8$	✓	✓	✓	✓	✓	✓

Table 2.2 Experimental conditions investigated.

2.4 Measurement techniques and operating procedures

In this section the measurement techniques are described, together with the steps taken to ensure the highest possible accuracy of measurement.

2.4.1 Flow visualisations

Visualisations of the secondary jets and their mixing with the mainstream flow were obtained by injecting smoke into the secondary air supply to the test cylinder. The smoke was created by a smoke generator which produced the smoke by heating a light oil in the presence of carbon dioxide. In order to improve the visibility of the smoke and the surface visualisations, the test cylinder and the tunnel side wall opposite the camera were covered with matt black 'Fablon'.

The surface flow visualisation material was a mix, by volume, of :-

45% paraffin

50% titanium dioxide

5% silicone oil

and a few drops of oleic acid.

Once the surface was coated with this material, new flow visualisation patterns could be obtained simply by wiping the surface with a paraffin soaked rag.

2.4.2 Static pressures

The static tapings on the test cylinder were connected via Norgren pressure switches to a Furness Controls Universal Pressure Anemometer, capable of measuring pressures in the range 0.1 to 2000 mm H₂O with an accuracy of $\pm 3\%$. In order to improve the accuracy of these measurements, the universal pressure anemometer was calibrated against a vernier scaled manometer, capable of measuring pressures to within ± 0.1 mm H₂O over the range 0 to 100 mm H₂O. A curve fit to this calibration has been used for the determination of pressures throughout this work and is accurate to $\pm 1\%$.

As a result of the unsteady nature of the flow, the output from the universal pressure anemometer was connected to a Solartron Time Domain Analyser to give the time meaned values of the measured pressures.

The upstream reference pressure was located on the tunnel centreline 4 cylinder diameters upstream of the centre of the cylinder. This point is within the potential field of the cylinder, and the static pressure was 0.03 dynamic heads above the unperturbed freestream value. The measured data have been corrected to account for this.

2.4.3 Velocities

Provided that it is within its range of calibration, a five-hole probe may be used to determine both the magnitude and direction of the velocity of the flow incident to it.

Referring to figure 2.5 for the definition of terms :-

$$\text{Velocity Magnitude} \quad |V| = f_m (p_o - p_m)$$

$$\text{Pitch Angle} \quad \alpha = f_p \left(\frac{p_3 - p_4}{p_o - p_m} \right)$$

$$\text{Yaw Angle} \quad \beta = f_y \left(\frac{p_1 - p_2}{p_o - p_m} \right)$$

$$\text{Total Pressure} \quad p_T = f_T (p_o - p_m) + p_o$$

$$\text{where} \quad p_m = \frac{p_1 + p_2 + p_3 + p_4}{4}$$

and the functional dependencies

f_m ; f_p ; f_y ; and f_T are determined by calibration.

(See Bryer and Pankhurst (1971)).

The probe calibration was carried out in a free jet normally used for the calibration of hot wire probes and covered a range of ± 45 degrees in both pitch and yaw at a flow velocity of 13 m/s.

Provided the Reynolds number of the flow around the probe is sufficiently high the calibration is independent of Reynolds number. Bryer and Pankhurst (1971) do not quantify the Reynolds number dependence of 5-hole probes, but they indicate that they became significant for sharp edged yawmeters below Reynolds numbers of 1500 (based on probe width). For the present measurements this corresponds to a flow velocity of about 8.2 m/s, which just falls within the range of measurements obtained (7.5 to 25 m/s) and suggests that, if there are any Reynolds number effects in the measurements, they are likely

to be small. This has been confirmed by analysis of calibration curves obtained at velocities of 6.5 m/s and 26 m/s.

The response time of the probe is dependent on the rate of flow through the small bore tubing of the probe and the volume of the pressure transducer and the connecting tubes. Since the probe is made as small as possible the response time can be significant. The arrangement used for the present work was chosen so as to reduce this response time to a minimum. Each tube of the probe was connected via a short length of 1.5 mm bore tube to a Norgren pressure switch, and thence by a further short length (32 cm) of the tubing to the universal pressure anemometer. This arrangement gave a response time of about 10 seconds to a change in conditions at the probe, and 6 seconds to a change of port in the pressure switch.

Since the probe is employed to determine the flow direction it is important to be able to position the probe with accuracy. This requires that the probe be positioned at the appropriate point on the cylinder corresponding to the start of each radial traverse, and that it be aligned parallel to the tangent to the surface of the cylinder at this point. This was achieved by scribing lines on the bottom floor of the windtunnel corresponding to the locus of each traverse and a normal to this line corresponding to the tangent to the cylinder. Using these lines, the probe could be positioned to within ± 0.3 mm of the desired location on the cylinder and to within ± 0.5 degrees of the tangent angle.

The determination of the calibration coefficients and the analysis of the results obtained by the five-hole probe were carried out by computer programs written specifically for the purpose.

2.4.4 Concentration

The validity of the use of techniques for determining film cooling effectiveness based on the analogy between heat and mass transfer has been demonstrated by several workers including, for example, Pedersen (1972) and Rastogi (1972). In the present work tracer gas concentration measurements were obtained using a Servomex Micro-Katharometer (M.K.158). The Katharometer consisted of two chambers containing heated wires which form two of the legs of a Wheatstone Bridge. Gas samples were drawn through each of the chambers, cooling the wires by an amount dependent on the thermal conductivity of the sample, and the flowrate and temperature of the gas. The sample for one chamber was drawn from the laboratory to form a datum, the other was drawn from the experiment. Provided the flowrate and temperature of the samples were identical then the Wheatstone Bridge imbalance arising from the difference in the temperature of the wires was proportional to the mass fraction of the tracer gas. To ensure that the temperature of both samples was identical, the Katharometer chambers were placed inside an oven to maintain a constant and uniform temperature environment. The flowrates of the samples were registered by rotameters and adjusted by needle valves to maintain constant and equal flowrates. The influence of the sampling rate on the accuracy of this equipment has been studied by several workers; for example Crabb (1979), and a sampling rate of 3ml/sec. has been

found to give accurate results without excessive testing time. This has been employed for the present work.

The secondary gas was seeded with the tracer gas (Helium) at the rate of approximately 1% by volume by injecting the helium at a constant rate at a point upstream of the restrictor (see figure 2.3). This ensured thorough mixing of the tracer gas with the secondary airflow in the plenum chamber.

To reduce errors arising from zero drift of the Katharometer and changes in the jet concentration, these were checked before and after each set of measurements. Additional checks on the zero drift of the Katharometer were made during the surface concentration measurements by taking a sample of laboratory air before sampling from each static.

2.5 Accuracy of the measurements

2.5.1 Static pressure

The total pressure of the windtunnel mainstream flow was constant to within $\pm 1\%$ of its nominal value and the secondary flowrate was maintained to within $\pm 1\%$ of the desired value. Whilst variations in the mainstream total pressure will be directly reflected in the static pressures, this is not the case for the variations in the secondary flow so that it is necessary to determine the exchange rate between the secondary flow and the static pressures. This may be found by studying the experimental results. In the worst case injection at $V_j / V_\infty = 1.8$ resulted in a perturbation of the unblown static pressure distribution of about a C_p of 0.8.

The perturbation of static pressure is approximately linearly dependent on the injection velocity so that a variation of $\pm 1\%$ in the secondary flow will lead to a variation in C_p of approximately ± 0.008 .

Inaccuracies arising from the static tappings themselves, i.e. burrs, leaks etc. would be readily apparent in the measured results for individual tappings and need not be considered here.

As described in section 2.4.2 the accuracy of the measuring equipment was $\pm 1\%$ so that the total error in the static pressure measurements was therefore :-

$$\begin{aligned}\Delta C_p &= \pm \sqrt{(0.01)^2 + (0.008)^2 + (0.01)^2} \\ &= \pm 0.016\end{aligned}$$

2.5.2 Velocity

We assume that the variations in the windtunnel and secondary flowrates translate directly into velocity, i.e.

$$\begin{aligned}\Delta V \text{ (Mainstream)} &= \pm 0.5\% \\ \Delta V \text{ (Secondary)} &= \pm 1.0\%\end{aligned}$$

The total error of the five-hole probe measurements is dependent on the errors incurred during the probe calibration as well as those arising in the course of the measurements. A major component of the calibration errors result from errors in recording pressures, and the calibration program has been employed to evaluate the influence of $\pm 1\%$ variations in pressures on the apparent velocity magnitude and direction. This has shown that the uncertainties in the velocity magnitude and direction arising from this cause are less than 2% and 0.6 degrees respectively.

Further errors in the calibration could arise from errors in the pitot probe measurement of the calibration jet velocity ($\pm 0.5\%$) and angular position (± 0.5 degrees). The total errors arising in the probe calibration are :-

$$\Delta \text{ Magnitude} = \sqrt{(2.0)^2 + (0.5)^2} = \pm 2.1\%$$

$$\Delta \text{ Direction} = \sqrt{(0.6)^2 + (0.5)^2} = \pm 0.8^\circ$$

Bryer and Pankhurst (1971) found that the effect of turbulence on five-hole probe measurements is dependent on its scale. For turbulence with small scale relative to the probe, the 5-hole probe reacts much as a pitot-static probe for which the total pressure registered is :-

$$\frac{P_T - P}{\frac{1}{2} \rho V_\infty^2} = 1 + \frac{k}{V_\infty^2}$$

and the static pressure exceeds the stream pressure by :-

$$\frac{P_{\text{meas}} - P}{\frac{1}{2} \rho V_\infty^2} = \frac{1}{2} \left(\frac{\overline{u'^2} + \overline{w'^2}}{V_\infty^2} \right)$$

Assuming isotropic turbulence this may be written in terms of k :-

$$\frac{P_{\text{meas}} - P}{\frac{1}{2} \rho V_\infty^2} = \frac{2/3 k}{V_\infty^2}$$

Downstream of a jet in crossflow the turbulence intensity frequently reaches 40% which is equivalent to $k/V_\infty^2 = 0.24$ for isotropic turbulence.

Assuming that all this energy lies in the small scales of turbulence, the error arising in the perceived pressures is therefore :-

$$\begin{aligned}\Delta p &= + \sqrt{1 + \frac{k}{V_\infty^2} - \frac{2}{3} \frac{k}{V_\infty^2}} - 1 \\ &= + \sqrt{1 + \frac{1}{3} \frac{k}{V_\infty^2}} - 1 \\ &= + 0.04\end{aligned}$$

Using the exchange rate between errors in pressure and velocity, already determined, the error in the velocity measurement is therefore :-

$$\Delta V = +8\%$$

The effects of large scale turbulence appear to reduce the apparent total pressure and hence tend to oppose the influence of the small scale turbulence, however, this is not sufficiently well understood to be able to quantify the effect.

As a result of the finite size of the probe, gradients in total pressure will give rise to apparent velocities normal to the stream. The magnitude of these apparent velocities will be dependent on the magnitude of the gradient and the size of the probe. Again, this effect is not thoroughly understood and cannot be quantified. It has been minimised in the present work by making the probe as small as possible and obtaining velocity measurements for the flow conditions with the largest scale features.

Reynolds number effects have been discussed in section 2.4.3 and may be neglected.

Leaks in any of the tubes connecting the probe to the universal pressure anemometer will not be readily apparent. Consequently, before taking each set of measurements, the probe was set facing upstream at a position well upstream of the cylinder in order to test for any change that could indicate a leak.

The errors arising from inaccuracies in recording pressures will be the same as those determined from this cause for the calibration, i.e.

$$\Delta \text{ Magnitude} = \pm 2\%$$

$$\Delta \text{ Direction} = \pm 0.6^\circ$$

Finally, using the lines scribed in the floor of the test section, the probe could be aligned to within ± 0.5 degrees.

In regions of low turbulence the total errors in the velocity measurements are therefore :-

$$\begin{aligned} \Delta \text{ Magnitude} &= \pm \sqrt{(0.5)^2 + (1.0)^2 + (2.1)^2 + (2)^2} \\ &= \pm 3.1\% \end{aligned}$$

$$\begin{aligned} \text{and } \Delta \text{ Direction} &= \pm \sqrt{(0.8)^2 + (0.6)^2 + (0.5)^2} \\ &= \pm 1.1 \text{ degrees} \end{aligned}$$

In regions of high turbulence intensity, such as in the wake of the film cooling jets, the error in the

velocity magnitude becomes :-

$$\begin{aligned}\Delta \text{ Magnitude} &= \sqrt{(3.1)^2 + (8.0)^2} \\ &= 8.6\%\end{aligned}$$

2.5.3 Concentration

For the concentration measurements we must account not only for the variations in the mainstream and secondary flow rates, but also for the flow variation of the tracer gas. This amounts to approximately $\pm 2\%$. The variation in the jet concentration is therefore :-

$$\begin{aligned}\Delta C_r &= \pm \sqrt{(1.0)^2 + (2.0)^2} \\ &= \pm 2.2\%\end{aligned}$$

Whilst the Time Domain Analyser was employed to obtain time mean pressure values for the static pressure and five-hole probe measurements, this was not available for most of the concentration measurements. The time mean concentration was therefore determined manually.

Subsequent checks using the Time Domain Analyser showed that the measurements obtained in this way were accurate to within $\pm 3\%$

Another source of error in the recording equipment was the zero drift. This could be equivalent to as much as ± 0.01 of the tracer gas concentration in the secondary flow and lead to difficulties in the measurement of low concentrations.

Bringing these sources of error together, the overall accuracy of the concentration measurements is :-

$$\begin{aligned}\Delta \left(\frac{c}{c_r} \right) &= \pm \sqrt{(2.2)^2 + (0.5)^2 + (3.0)^2 + (1.0)^2} \\ &= \pm 4\%\end{aligned}$$

CHAPTER 3

PRESENTATION AND DISCUSSION OF THE MEASUREMENTS

3.1 Introduction

The apparatus and techniques employed for the measurements have been described in chapter 2. In this chapter the results obtained during the experiment are presented and discussed. Measurements for the cylinder without secondary air ejection are described in section 3.2, and those for the three configurations with secondary air ejection ($\theta_s = 0, \pm 20$ and ± 40 degrees) are discussed in sections 3.3, 3.4 and 3.5 respectively. Concluding remarks are contained in section 3.6.

3.2 Results without secondary air ejection

Measurements were made without secondary flow to check the symmetry and two-dimensionality of the flow around the cylinder, and also to provide confirmation of the accuracy of the five-hole probe measurements.

The measured static pressure distribution around the cylinder is shown in figure 3.1. Results obtained on the opposite side of the stagnation line ($\theta < 0$) are in close agreement and indicate the symmetry of the flow. Confirmation of the accuracy of the measurements has been sought by completing a potential flow analysis of the flow around the cylinder. The solution employs the method of singularities and includes the effect of the windtunnel walls. A full description of the method is given in appendix I. As a result of viscous effects the measured static pressure distribution will not be in perfect agreement with that given by potential flow theory. Similar

comparisons for cylinders in unbounded crossflows would suggest however that good agreement should be obtained for $\Theta < 30$ degrees, see Schlichting (1968) for example. As shown in figure 3.1, the present static pressure measurements are in excellent agreement with the theoretical result for $\Theta < 40$ degrees.

Profiles of the tangential (V_θ/V_∞) and radial (V_r/V_∞) velocity components derived from five-hole probe measurements are shown in figure 3.2, together with velocity profiles predicted by the potential flow analysis. The agreement obtained is generally consistent with the neglect of viscous effects in the prediction and the experimental errors determined in section 2.5.2. The measurements show however that the point adjacent to the wall has been influenced by the presence of the surface and an apparent velocity component away from the wall may be observed in the profiles of V_r/V_∞ . Examination of the V_z/V_∞ profiles allows the two-dimensionality of the flow to be determined. The results show that $|V_z/V_\infty| \leq 0.02$ which corresponds to flow deviations of less than 1.2 degrees and is not significantly greater than the experimental error. We conclude that the flow in the mid-span plane of the cylinder is two-dimensional.

3.3 Single row ; $\Theta_s = 0$ degrees

3.3.1 Flow visualisations

Smoke and surface flow visualisation pictures for this configuration are shown in figure 3.3. The results indicate the strong effect of the velocity ratio (V_s/V_∞) on the flow. For $V_s/V_\infty > 1$, the flow becomes bistable and, with the splitter plate fitted, the ejected flow switches from one

side of the cylinder to the other. With the splitter plate removed the flow is stable, but not symmetrical about the nominal stagnation point, the jets flowing over only one of the surfaces of the cylinder as shown in figures 3.3(a) and (b). (Note that in the surface visualisation 3.3(a) the bottom jet is flowing over the near side of the cylinder, causing the feature observed towards the base). The cause of the bistable flow has been deduced from predictions presented in chapter 7 and is a result of the shedding of vortices generated along the front of the cylinder by the ejection, which then interact with those at the rear of the cylinder. Further discussion of the bistability is given in section 7.3.1.

For $V_r/V_\infty < 1$, the secondary jets divide to flow symmetrically to either side of the stagnation point, figures 3.3(c) and (e), but now there is a tendency for adjacent jets to pair as shown in figure 3.3(d). Crabb (1979) has observed similar behaviour in a study of a row of jets in crossflow. Where there is a significant mainstream crossflow at the jet exit, however, the mainstream flow penetrating between the jets inhibits the pairing and Crabb (1979) only observed pairing where one of the jets was located asymmetrically between the adjacent jets. For the present ejection configuration the mainstream crossflow is very weak so that the pairing of the jets is not inhibited. The random nature of the pairing suggests that it is triggered by perturbations in the mainstream flow causing adjacent jets to move together. Provided the perturbations are sufficiently large, the jets will be drawn together by mutual entrainment. Separation of the jet pair arises as a result of subsequent perturbations of the mainstream flow.

3.3.2 Static pressure distribution

As a result of the bistability of the flow, the only measurement obtained for $J > 1$ was the static pressure distribution, with the splitter plate removed, for $V_x/V_\infty = 2.0$. The results, shown in figure 3.4 indicate the asymmetry of the flow, the stagnation line lying between 5 and 10 degrees from its nominal position.

3.3.3 Velocity profiles

Velocity profile measurements were obtained for $J = 0.9$ the results for $z/D = 0.0$ and 1.0 being shown in figures 3.5(a) and (b) respectively. It is notable that the ejection of secondary fluid has had little effect on the velocity field, the V_θ and V_r components being only slightly different from those obtained without ejection, shown in figure 3.2. This is not typical of jets in crossflow in which the secondary jet causes large distortions of the velocity profiles downstream of the ejection point. The distortions are largely a result of the blockage of the mainstream flow caused by the secondary jets. For ejection along the stagnation line however, the cross-flow component of the mainstream is negligible, and the blockage caused by the secondary jets must be seen in relation to that of the cylinder. Since in the present case $R/D = 6.25$, the blockage caused by the cylinder must dominate that of the secondary jets, the effect of ejection on the velocity field is relatively small.

The radial and lateral velocity components (V_r/V_∞) and (V_z/V_∞) show evidence of a "streamwise" vortex. These vortices are formed at the ejection point and pass over the surface of the cylinder to either side of the stagnation

point. Each vortex is therefore continuous from the $\Theta > 0$ separation point to the leading edge and on around to the $\Theta < 0$ separation point. Further analysis of the vortex structure is made difficult by a slight asymmetry of the flow, causing lateral velocity components of up to $V_z/V_\infty = 0.06$ at $z/D = 0.0$. Since the measurements are mean values obtained over a long period of time, this could possibly arise as a result of a slight bias in the jet pairing. It is unlikely to have a significant influence on the basic features observed here.

3.3.4 Concentration measurements

Measurements of tracer gas concentration for $J = 0.89$ and 0.48 are shown in figures 3.6 to 3.9. The concentration profiles presented in figures 3.6 and 3.8 show that the secondary fluid penetrates to $r/R \approx 1.5$, i.e. about 3 hole diameters from the surface for $V_z/V_\infty = 0.89$ and about 1 hole diameter from the surface for $V_z/V_\infty = 0.48$. At both conditions the jet penetration remains approximately constant with increasing Θ . This is because the action of entrainment and diffusion which tend to spread the secondary jet fluid away from the wall is balanced by the effect of the acceleration which draws the fluid towards the wall. The peaks in the profiles at $\Theta = 0$ and 26.6 degrees indicate that the jet is detached from the surface. Beyond $\Theta = 45$ degrees the acceleration has caused the reattachment of the jets and no peaks may be observed in the profiles.

The surface concentration ($\eta_{i\omega}$) measurements for $J = 0.99$ and 0.48 , presented in figures 3.7 and 3.9, display significant differences. For example; the measurements for $J = 0.99$

indicate very strong lateral spreading of the secondary fluid, whereas those for $J = 0.48$ show evidence of little lateral spread. The strong lateral spreading for $J = 0.99$ is partly caused by the pairing of the secondary jets observed at this condition in the flow visualisations (c.f. section 3.3.1), and partly a reflection of the large size of the streamwise vortex set up by the secondary injection. Indeed, the streamwise vortex is so effective at transporting secondary fluid away from the jet centreline, that the measurements show lower values of η_{is} at $z/D = 0.0$ than at $z/D > 0.0$. For $J = 0.48$ the streamwise vortices are relatively small, and jet pairing does not occur at this condition so that the lateral spread is correspondingly much smaller than that for $J = 0.99$. The reduced penetration and lateral spread of secondary fluid for the lower velocity ratio condition causes a confinement of the secondary fluid which results in higher levels of surface concentration at $z/D = 0.0$ and 0.5 .

3.4 Two rows : $\Theta_r = \pm 20^\circ$

3.4.1 Flow visualisations

The flow visualisation results for $\Theta_r = \pm 20^\circ$ are presented in figure 3.10. The large scale interactions with the mainstream flow observed for the $\Theta_r = 0^\circ$ configuration do not occur and the flowfield is more akin to that normally observed for jets in cross-flow. The smoke flow visualisations show that the penetration and lateral spread of the secondary fluid is considerably less than that observed for the $\Theta_r = 0^\circ$ configuration. This reflects the action of the mainstream cross-flow which causes the secondary jets to bend towards the surface, as described in section 1.3.2.

The surface flow visualisations presented in figure 3.10 show a marked effect of the ejection. For $J = 2.0$ the surface fluid is drawn around the periphery of the hole into the low pressure region downstream of the jet. Beyond this region, which extends approximately $1.D$ downstream of the hole, the convergence of the visualisation material towards the jet centreline indicates the action of the streamwise vortices, causing the entrainment of mainstream fluid under the secondary jets. At the lower jet velocity ratios the effects are much less marked and, at the lowest velocity ratio, there is no sign of the entrainment of mainstream flow under the jet, indicating that the jets are attached to the surface.

At all three velocity ratios a small region of reverse flow may be observed at the downstream edge of the holes, into which the flow visualisation material has collected. It is impossible to determine the size of the reverse flow regions because of the effect of gravity on the collected material.

3.4.2 Velocity profiles

Velocity profile measurements were obtained for $J = 1.98$, the results for $z/D = 0.0$ and 1.0 being shown in figures 3.11(a) and (b) respectively. The influence of the ejection on the tangential velocity component (v_e/v_∞) is similar to that observed in the streamwise velocity component measurements of Ramsey and Goldstein (1971) and Crabb (1979) on rows of holes and jets in crossflow on a flat plate. Although in the present work the jet-mainstream interaction will be influenced by the surface curvature and the acceleration of the mainstream flow, this suggests that, unlike the $\theta_f = 0^\circ$ configuration, the interaction is essentially similar to that

of the jets in crossflow.

If instead of characterising the velocity of the injected flow by the unperturbed velocity of the mainstream approaching the cylinder (V_∞) we chose the peak V_θ velocity component near the wall in the vicinity of the ejection point for the unblown case (V_L), the effective velocity ratio becomes $J_L = 2.66$. This definition is more consistent with that employed in flat plate studies and permits comparison between the present measurements and published flat plate results. The data of Crabb (1979) for a row of holes with $P/D = 4$ and $J = 2.3$ is perhaps the most appropriate. The tangential velocity profile (V_θ/V_∞) for $\Theta = 36.9$ degrees and $z/D = 0.0$ plotted in figure 3.11(a) displays a peak at $r/R = 1.25$ (1.5D from the surface). In terms of surface perimeter downstream of the hole this profile corresponds approximately with that measured by Crabb (1979) at $X/D = 2$ for which the peak velocity occurs 3D from the surface. Comparisons with the results at $\Theta = 45$ and 63.4 degrees also show much decreased jet penetration with ejection from the cylinder compared with the flat plate results. This reflects the action of the acceleration of the flow around the cylinder which gives rise to significant radial velocity components towards the surface countering the momentum of the ejected fluid and reducing the jet penetration. This outweighs the influence of the surface curvature which, as Bradshaw (1973) observes for wall jets on convex surfaces, leads to increased spreading of the jet due to enhanced turbulent mixing.

The V_r and V_z velocity profiles shown in figure 3.11 indicate the presence of a strong streamwise vortex near the surface of the cylinder. This corresponds with one of the

contra-rotating vortex pair characteristic of the flow downstream of jets in crossflow. Inspection of the V_z -profile for $z/D = 1.0$ shows that the centre of the vortex lies at $r/R = 1.15$ at all three traverse locations, implying no increase in the size of the vortex. It can also be seen that the strength of the vortex, inferred from the velocity near the surface, decreases by a factor of 2 between $\Theta = 36.9$ and 63.4 degrees. This arises partly as a result of the dissipation of the angular momentum of the vortex by shear stresses, and partly as a result of the entrainment of mainstream fluid into the vortex induced by the acceleration.

The V_z/V_∞ profile at $\Theta = 36.9$ degrees and $z/D = 0$ indicates a small region of spanwise flow in the vicinity of the low tangential (V_θ/V_∞) velocities. This region, if not part of an area of reversed flow, is just downstream of flow reversal and the measurements will be subject to probe interference. The existence of a small asymmetry in the flow is confirmed however by the profiles at $\Theta = 45$ degrees. The lack of symmetry is unlikely to have a large effect on the major features of the flow because it is of limited extent and the deviation associated with it is only about 5 degrees.

3.4.3 Concentration measurements

Measurements of tracer gas concentration for this configuration are shown in figures 3.12 to 3.17. Concentration profiles for $J = 1.98$, 1.11 and 0.55 are shown in figures 3.12, 3.14 and 3.16 respectively. As with the V_θ/V_∞ velocity profiles presented in section 3.4.2 the profiles are of similar form to the scalar profiles obtained by ejection into zero pressure gradient flow over a flat plate (e.g. Ramsey and

Goldstein (1971) and Crabb (1979), but indicate a reduced penetration of the secondary jets. The results for $J = 1.98$ and 1.11 display peaks above the surface of the cylinder indicating that the secondary jet is detached from the surface of the cylinder, allowing the intrusion of the mainstream beneath the jet. The measurements for $J = 0.55$ peak above the surface only at $\Theta = 26.6$ degrees and it would appear that by $\Theta = 45$ degrees the jet has become attached to the surface. The flow visualisation results presented in section 3.4.1 show that, in fact, the jet does not detach from the surface, and the peaks in the concentration profile at $\Theta = 26.6$ degrees may be the result of probe interference.

The surface concentration measurements for $J = 1.98$, 1.11 and 0.55 presented in figures 3.13, 3.15 and 3.17 respectively show the lack of lateral spread of the secondary fluid observed in the flow visualisations (c.f. section 3.4.1). Virtually no secondary fluid is detected at $Z/D = 1.0$ upstream of the separation of the mainstream from the cylinder at $\Theta \sim 90$ degrees. The lateral spreading of the secondary fluid is largely caused by the action of the streamwise vortices associated with the ejection and the results reflect the small scale of the vortices set up by the ejection from the cylinder. The results also show that η_{10} increases significantly as the injection rate is decreased. This reflects the decreased penetration of the ejected flow and the reduction, or complete suppression of the intrusion of mainstream flow beneath the jets.

3.5 Two rows : $\Theta_{\pm} = \pm 40$ degrees

3.5.1 Flow visualisations

Flow visualisations for $\Theta_{\pm} = \pm 40$ degrees are shown in figure 3.18. Some of the features of the flow visualisations

are similar to those observed for the $\Theta_s = \pm 20$ degree configuration : the small lateral spreading of the secondary fluid for example, and the reverse flow at the downstream edge of the holes. One of the most marked features of the surface visualisations, which was not apparent for the previous configuration, is the line of separation denoted by the boundary between the light and dark regions. The result for $J = 1.0$ shows clearly that the separation originates from upstream of each hole. Foss (1980) has observed separation of the mainstream upstream of a jet in crossflow, the large scale of his experiment allowing the full complexities of the flow to be determined. In the simplest case a pair of contra-rotating vortices were observed immediately upstream of the hole. The vortex associated with the separation upstream of the hole is akin to a "horseshoe" vortex and occurs because the near-wall fluid of the boundary layer approaching the jet is unable to withstand the adverse pressure gradient upstream of the hole. The second vortex lies nearer the jet and is caused by the shear between the jet and the surrounding fluid. This provides reasonable grounds for the postulation of the existence of two pairs of contra-rotating vortices downstream of each hole. A larger pair of vortices related to those normally associated with jets in crossflow which originate from the shear generated vortex with a smaller pair of vortices lying beneath them rotating in the opposite sense which originate from the "horseshoe" vortex. The separation line downstream of the holes, shown in the surface flow visualisations, is associated with the upwash region between the smaller and larger vortices.

3.5.2 Velocity profiles

Measurements of the velocity profiles for $J = 1.8$ are shown in figure 3.19. The major features of the results are similar to those of the $\Theta_j = \pm 20$ degree configuration except that the peaks in the velocity profiles are more pronounced and the symmetry about $Z/D = 0.0$ is improved. No measurements could be obtained near the wall for $\Theta = 53.1$ degrees and $Z/D = 0.0$ because of flow reversal. No evidence for the existence of the smaller pair of contra-rotating vortices, postulated in the discussion of the flow visualisation results (c.f. section 3.5.1), may be seen in the measurements and it is probable that they are too small to be resolved by the five hole probe.

In relation to the results for $\Theta_j = \pm 20$ degrees, the measurements for this configuration show reduced penetration of the secondary jets. This reflects the increase in the velocity V_L (c.f. section 3.4.2) between $\Theta = 20$ and 40 degrees resulting in a decrease in the local velocity ratio J_L .

3.5.3 Concentration measurements

Concentration profiles for $J = 1.8, 1.0$ and 0.5 are shown in figures 3.20, 3.22 and 3.24 respectively. It is useful to compare the results for the ± 40 degree configuration with those of the ± 20 degree configuration on the basis of the local velocity ratio, J_L :

$\Theta_j = \pm 20^\circ$		$\Theta_j = \pm 40^\circ$	
J	J_L	J	J_L
1.98	2.6	1.80	1.30
1.11	1.45	1.00	0.77
0.55	0.73	0.50	0.39

In terms of J_L the $J = 1.80$ and 1.00 conditions of the $\Theta_s = \pm 40$ degree configuration relate to the $J = 1.11$ and 0.55 conditions of the $\Theta_s = \pm 20$ degree configuration. Comparing figure 3.20 with 3.14 and figure 3.22 with 3.16 it can be seen that the concentration profiles for the two configurations are in approximate correspondence. The agreement between the surface concentration results is poorer however (compare figure 3.21 with 3.15 and figure 3.23 with 3.17). This is largely because of the existence of the radial velocity components and because V_L relates to the peak V_θ component near the wall, not the velocity further from the surface.

Concentration profiles for $J = 0.5$ are shown in figure 3.24 and it can be seen that the penetration of the secondary fluid is very small (approximately $0.5 D$) and that the jets remain attached to the cylinder. The surface concentration measurements presented in figure 3.25 show a peak level of $\eta_w > 0.9$ and a steady decrease with increasing Θ . This is due to a combination of the low massflux of the secondary fluid and the rapid diffusion of the secondary fluid into the mainstream as a result of the high concentration on the surface and the thinness of the layer.

3.6 Concluding remarks

The results presented in this chapter have shown that the flowfield set up by ejection from a row of holes on the stagnation line has a different character to the other ejection geometries studied. The difference in character of the stagnation line ejection is a result of a combination of the lack of mainstream crossflow, and a large scale interaction between the secondary fluid and the mainstream which influences the location of the

stagnation line. For $J \geq 1.0$ the flow is bistable and symmetry about the nominal stagnation line cannot be obtained.

Neither of these features are present for the $\Theta_s = \pm 20$ and ± 40 degree configurations which are essentially similar in character to jets in crossflow. The velocity ratio (V_s/V_L) is consistent with the definition employed for jets in crossflow and allows comparisons to be made between the present results and previous measurements of jets in crossflow. The comparisons show that the penetration of the secondary jets is much smaller in the present results largely as result of radial velocity components in the mainstream flow.

For the purposes of turbine blade cooling the influence of the ejected flow on the aerodynamic efficiency and the heat transfer to the surface of the component are of prime interest. The following paragraphs describe the aerodynamic and "heat" transfer aspect of the present results in the context of the cylinder. The implications for turbine blade cooling are considered in chapter 8,

For $J < 1$ the ejection on the stagnation line has little influence on the aerodynamics of the mainstream flow, the velocity profiles showing little change from those obtained without ejection. Indeed the profiles of total pressure obtained from the five hole probe measurements (not presented) show that the ejection produces very small mixing losses. In contrast, for $J > 1$ the ejection has a major influence on the flow around the cylinder, which becomes asymmetric about the nominal stagnation point. The other two ejection configurations examined show a much greater effect of the ejection on the velocity profiles, and the total pressure measurements suggest that the aerodynamic

losses due to turbulent mixing increase as Θ_J becomes larger.

The impervious wall effectiveness results presented in the previous sections show trends of increasing effectiveness along the jet centreline ($Z/D = 0.0$) with increase of Θ_J and decrease of J . In the examination of these results however, it is important to bear in mind that one of the requirements for turbine blade cooling design is to seek configurations giving the maximum overall cooling effect for the minimum coolant flow. Hence it is useful to compare the three configurations on the basis of the overall cooling effect produced for a given total coolant flow. An examination of the results for the $J = 0.90, 0.55$ and 0.50 conditions of the $\Theta_J = 0, \pm 20$ and ± 40 degree configurations respectively, for which the total flow of coolant is approximately the same, shows that the greatest coverage of the surface is given by the $\Theta_J = 0$ configuration, but the highest levels of η_{iw} are obtained for the $\Theta_J = \pm 40$ degree configuration. The overall cooling effect given by the 3 configurations has been studied by estimating the area mean values of η_{iw} over the surface for Θ between 0 and 80 degrees, with the following results :-

$\Theta_J =$	0 degrees,	η_{iw} (Area mean)	= 0.045
	= ± 20 degrees		= 0.105
	= ± 40 degrees		= 0.148

It can be seen that in spite of leaving the first forty degrees of the surface uncooled the ± 40 degree configuration gives by far the greatest cooling, as a result of the higher levels of η_{iw} attained.

CHAPTER 4

THEORETICAL CONSIDERATIONS

4.1 The governing equations

The equations governing the transport of mass momentum and energy of a fluid may be written, using Cartesian tensor notation :-

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho V_i}{\partial x_i} = 0 \quad (4-1)$$

$$\rho \frac{\partial V_i}{\partial t} + \rho V_j \frac{\partial V_i}{\partial x_j} = \rho F_i - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial V_k}{\partial x_k} \delta_{ij} \right] \quad (4-2)$$

$$\rho \frac{\partial h}{\partial t} + \rho V_j \frac{\partial h}{\partial x_j} = \rho V_j F_j + \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left[V_k \mu \left(\frac{\partial V_k}{\partial x_j} + \frac{\partial V_j}{\partial x_k} \right) - \frac{2}{3} \mu \frac{\partial V_l}{\partial x_l} V_k \delta_{jk} \right] \quad (4-3)$$

where

h is the total specific enthalpy defined by

$$h = h_{ref} + \int_{T_{ref}}^T c_p dT + \frac{1}{2} V_j V_j$$

C_p specific heat at constant pressure

The density of the fluid is obtained from an equation of state; e.g.

$$\rho = \rho(p, T) \quad (4-4)$$

Equations (4-1) to (4-4) form a closed set of six equations in the six unknowns V_1, V_2, V_3, p, T and ρ so that in principle it is possible to solve the equation set to determine the values of the unknowns. In practise the equations are difficult to solve because of the non-linearity of the convection terms on the left hand side and the strong coupling between them.

For the purposes of the present work some simplification of the equations is permissible, since all the work presented in chapters 5, 6 and 7 concerns steady, low velocity flow, with small temperature differences. The maximum Mach and Eckert numbers that occur are 0.2 and 0.15 respectively. Thus it is possible to neglect the influence of pressure on the density of the fluid and the kinetic energy and shear work terms may be dropped from the energy equation, which may be further simplified by assuming a constant specific heat. Finally the only external body forces occurring arise from gravitationally driven buoyancy effects. These forces are several orders of magnitude smaller than those generated by the pressure field and may safely be neglected.

The simplified equations may be written :-

$$\frac{\partial \rho V_i}{\partial x_i} = 0 \quad (4-5)$$

$$\rho V_j \frac{\partial V_i}{\partial x_j} = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial V_l}{\partial x_l} \delta_{ij} \right] \quad (4-6)$$

$$\rho V_j \frac{\partial T}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\frac{k}{c_p} \frac{\partial T}{\partial x_j} \right) \quad (4-7)$$

$$\rho = \rho(T) \quad (4-8)$$

4.2 The equations of motion for turbulent flow

The equations (4-5) to (4-8), which are written in terms of instantaneous values of the dependent variables are applicable to both turbulent and non-turbulent flows. In order to obtain a more tractable form of the equations for turbulent flow we employ the technique of time averaging introduced by Reynolds. We start by representing the instantaneous value of a quantity by the sum of the mean and unsteady components: i.e.

$$\left. \begin{aligned} \rho &= \bar{\rho} + \rho' \\ v_i &= \bar{v}_i + v'_i \\ p &= \bar{p} + p' \\ T &= \bar{T} + T' \end{aligned} \right\} \quad (4-9)$$

Where

$$\left. \begin{aligned} \bar{\rho} &= \frac{1}{\tau_i} \int_t^{t+\tau_i} \rho \, d\tau ; \quad \bar{v}_i = \frac{1}{\tau_i} \int_t^{t+\tau_i} v_i \, d\tau ; \\ \bar{p} &= \frac{1}{\tau_i} \int_t^{t+\tau_i} p \, d\tau ; \quad \bar{T} = \frac{1}{\tau_i} \int_t^{t+\tau_i} T \, d\tau \end{aligned} \right\} \quad (4-10)$$

$$\text{and } \int_t^{t+\tau_i} \rho' \, dt = \int_t^{t+\tau_i} v'_i \, dt = \int_t^{t+\tau_i} p' \, dt = \int_t^{t+\tau_i} T' \, dt = 0 \quad (4-11)$$

The time interval of the integration, τ_i , is chosen such that it is small compared with the global time scale of the flow and yet large compared with the time scale of the turbulent motions. The molecular viscosity and thermal conductivity of most fluids are strong functions of temperature so that these too will have mean and fluctuating components.

In turbulent flows, however, the influence of molecular quantities is generally relatively small. In the regions of the flow where molecular action is significant the turbulent fluctuations are suppressed. In either case the fluctuations of molecular viscosity and thermal conductivity may be neglected.

Substituting equations (4-9) into equations (4-5), (4-6) and (4-7) and time averaging, we obtain an equation set written in terms of mean values and correlations of the fluctuating quantities. As discussed by Hinze (1975), provided $\rho'/\bar{\rho} \ll 1$ correlations involving ρ' may be neglected. This condition is satisfied in the present study since we are dealing with low Mach number flows with small temperature differentials. Finally we obtain the equations :-

$$\frac{\partial \bar{\rho} \bar{V}_i}{\partial x_i} = 0 \quad (4-12)$$

$$\bar{\rho} \bar{V}_j \frac{\partial \bar{V}_i}{\partial x_j} = - \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) - \frac{2}{3} \mu \delta_{ij} \frac{\partial \bar{V}_k}{\partial x_k} - \bar{\rho} \overline{V'_i V'_j} \right] \quad (4-13)$$

$$\bar{\rho} \bar{V}_j \frac{\partial \bar{T}}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\frac{k}{c_p} \frac{\partial \bar{T}}{\partial x_j} \right) - \bar{\rho} \overline{V'_j T'} \right] \quad (4-14)$$

The Reynolds stress and turbulent heat flux terms $-\bar{\rho} \overline{V'_i V'_j}$ and $-\bar{\rho} \overline{V'_j T'}$ must be defined by additional equations to allow closure of the equation set. Launder and Spalding (1972) consider a wide range of methods of modelling these terms, assessing the models in terms of accuracy, economy and the ability to represent a wide range of flows.

4.3 Turbulence modelling

4.3.1 Choice of turbulence model

The simplest turbulence models determine the turbulence correlations in terms of local quantities. Perhaps the most widely used model of this type is the mixing length model due to Prandtl (1925) in which a turbulent viscosity is defined in terms of an empirically specified mixing length and the local velocity gradient. The model has been extensively employed in the calculation of boundary layer flows for which a simple mixing length distribution may be employed. It is unable to cope with more complex flows, however, where the turbulence is no longer in equilibrium (Production $\neq$ Dissipation) and cannot be characterised by local quantities. To a certain extent this limitation is removed in one-equation turbulence models, in which a turbulent viscosity is related to an algebraically defined length scale, and the kinetic energy of turbulence, k , determined from the solution of a transport equation for k . The models of Glushko (1965) and Kolmogorov (1942) are of this form.

The two equation turbulence model is the simplest form of turbulence model in which the turbulent correlations are uniquely determined by the solution of transport equations. For recirculating flows, in which the turbulence is not in equilibrium and a length scale cannot be algebraically defined, the two-equation turbulence model is the simplest form that can be employed. The k - ϵ model employed throughout this study is of this type. However, in reality, and because it has an eddy viscosity form, this is also restricted to near-equilibrium flows. In this model transport equations for the turbulence kinetic energy (k) and turbulent energy dissipation rate (ϵ) are solved, the

closure being completed by the expressions :-

$$\mu_t = \rho c_\mu k^2 / \epsilon \quad (4-15)$$

$$\bar{\rho} \overline{V_i' V_j'} = \frac{2}{3} \bar{\rho} k \delta_{ij} - \mu_t \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) \quad (4-16)$$

$$\bar{\rho} \overline{V_j' T'} = - \frac{\mu_t}{\sigma_t} \frac{\partial \bar{T}}{\partial x_j} \quad (4-17)$$

This model has been employed by many authors in the study of two and three-dimensional shear flows including recirculation. (E.g. Jones and Launder (1972), Vasilic-Melling (1977)). The $k-\epsilon$ model has also been employed in previous studies of slots and jets in crossflow (e.g. Matthews (1972), Patankar, Basu and Alpay (1977), and Crabb (1979)).

As discussed by Crabb (1979) the use of a turbulence model based on an isotropic turbulent viscosity may be inappropriate for the present problem. An alternative approach is to model the transport equations for the Reynolds stresses and turbulent heat fluxes directly. In three-dimensional recirculating flows with heat transfer this involves the solution of equations for the six Reynolds stresses and three turbulent heat fluxes. These equations include triple correlations of the fluctuating quantities (sixteen in all) and correlations between pressure fluctuations and velocity or temperature fluctuations. All of these terms require modelling in order to close the equation set. In view of this it is not surprising that such models have not yet been employed for flows of the present type, however, studies have been completed using Reynolds stress models for simpler flow problems.

For example in the computation of two-dimensional channel flows Hanjalic and Launder (1972) employed a turbulence model based on the solution of an equation for the turbulent kinetic energy and one of the turbulent stresses. For jets and wake flows Launder, Reece and Rodi (1975) solved five of the Reynolds stress equations. More recently turbulence models have been developed in which the Reynolds stresses are determined from the combination of algebraic expressions for the correlation coefficients $\left(\overline{v_i'v_j'} / k \right)$ and the solution of an equation representing the transport of turbulent energy, k (See Rodi (1976) and Gibson (1978)).

Turbulence models based on the direct calculation of Reynolds stresses show considerable promise in the treatment of flows with buoyancy or streamline curvature, however, much work remains to be done to establish the assumptions on which the models are based. For example in the computation of the flow around bluff bodies Pope and Whitelaw (1976) found little difference between the results obtained with a two equation turbulence model and Reynolds-stress models.

In relation to the present study, the additional complexity and computational requirements of a Reynolds-stress turbulence model would not appear to be justified. Consequently the simpler two-equation $k-\epsilon$ model has been employed throughout.

4.3.2 The $k-\epsilon$ turbulence model

Equations representing the transport of turbulence kinetic energy (k) and energy dissipation rate (ϵ) may be derived from the Navier-Stokes equations (See Hinze (1975), and Harlow and Nakayama (1968)). Several of the terms in the resulting equations cannot be readily evaluated and require modelling.

A full description of the modelling process may be found in Priddin (1975). The resulting model equations are :-

$$\bar{\rho} \bar{V}_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_c}{\sigma_c^k} \right) \frac{\partial k}{\partial x_j} \right] + G - \bar{\rho} \epsilon \quad (4-18)$$

$$\bar{\rho} \bar{V}_j \frac{\partial \epsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_c}{\sigma_c^\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + c_1 \frac{\epsilon}{k} G - c_2 \bar{\rho} \frac{\epsilon^2}{k} \quad (4-19)$$

$$\text{Where } G = \left[\mu_c \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \bar{\rho} k \right] \frac{\partial V_i}{\partial x_j}$$

The six empirical constants take the values :

$$C_\mu \text{ (Eqn. 4-15)} = 0.09$$

$$c_1 = 1.44$$

$$c_2 = 1.92$$

$$\sigma_c^k = 1.0$$

$$\sigma_c^\epsilon = 1.3$$

$$\sigma_c^T \text{ (Eqn. 4-17)} = 0.9$$

These values have been chosen following extensive numerical experiment to give good agreement over a wide range of flows (Launder and Spalding (1972)).

4.4 Numerical solution of the equations

The two-dimensional and three-dimensional computations presented in the following chapters are completed using different numerical methods. The two dimensional calculations are completed using the Imperial College TEACH code, Gosman and Pun (1974), whilst a programme written by Dr. W.P. Jones for Rolls Royce Ltd. is employed for the three-dimensional calculations. Although the methods differ in detail the essential features described here are relevant to both. The three-dimensional numerical method has been written to allow computation on either rectangular Cartesian or cylindrical polar meshes. For simplicity, the following description is based on the rectangular Cartesian mesh form.

4.4.1 The finite difference equations

We begin by noting that it is possible to write the equations solved (4-12), (4-13), (4-14), (4-18) and (4-19) in the general form :-

$$\frac{\partial}{\partial x}(\rho V_x \phi) + \frac{\partial}{\partial y}(\rho V_y \phi) + \frac{\partial}{\partial z}(\rho V_z \phi) =$$

$$\frac{\partial}{\partial x}\left(\Gamma \frac{\partial \phi}{\partial x}\right) + \frac{\partial}{\partial y}\left(\Gamma \frac{\partial \phi}{\partial y}\right) + \frac{\partial}{\partial z}\left(\Gamma \frac{\partial \phi}{\partial z}\right) + S^\phi \quad (4-20)$$

where ϕ may stand for $1, V_x, V_y, V_z, T, k$ or ϵ and overbars have been dropped for convenience.

In order to derive the finite difference equations we make use of the staggered grid arrangement shown in figure 4.1 which was first employed by Harlow and Welch (1965).

The grid is arranged so that the velocities are located halfway between the pressures that drive them. Such an arrangement requires the adoption of different finite difference control volumes for each of the velocity components.

Consider the transport equation for a conserved scalar, ϕ , in which $S^\phi = 0$. Assuming a linear variation of fluid properties between the nodal points, and that the flux of a quantity through the face of a control volume is constant across that face, then it is possible to integrate equation (4-20) over the control volume to obtain an algebraic equivalent of the conservation equation.

$$\begin{aligned}
 \int_V (flux) dV = & -(\rho V_y \Delta x \Delta z)_n [f_n \phi_p + (1 - f_n) \phi_N] \\
 & + (\rho V_y \Delta x \Delta z)_s [f_s \phi_S + (1 - f_s) \phi_p] \\
 & - (\rho V_z \Delta x \Delta y)_e [f_e \phi_p + (1 - f_e) \phi_E] \\
 & + (\rho V_z \Delta x \Delta y)_w [f_w \phi_w + (1 - f_w) \phi_p] \\
 & - (\rho V_x \Delta y \Delta z)_r [f_r \phi_p + (1 - f_r) \phi_R] \\
 & + (\rho V_x \Delta y \Delta z)_l [f_l \phi_L + (1 - f_l) \phi_p]
 \end{aligned}
 \tag{4-21}$$

The weighting functions $f_n, f_s, f_e, f_w, f_r, f_c$ are determined from consideration of the equation representing the combined one dimensional convection and diffusion flux through the face of the control volume. Consider the West-East flux through the control volume :

$$\rho V_z \frac{\partial \phi}{\partial z} - \frac{\partial}{\partial z} \left(\Gamma \frac{\partial \phi}{\partial z} \right) = 0 \quad (4-22)$$

Assuming uniform properties the exact solution of this equation at the Western face of the control volume gives :-

$$(\text{Total flux})_w = (\rho V_z \Delta x \Delta y)_w [f_w \phi_w + (1-f_w) \phi_p] \quad (4-23)$$

$$\text{Where } f_w = \exp(Pe_w) / [\exp(Pe_w) - 1] \quad (4-24)$$

$$\text{and } Pe_w = (\rho V_z)_w (z_p - z_w) / \Gamma_w \quad (4-25)$$

Since the computation of exponentials is expensive the weighting factors are approximated, with little loss of accuracy by the expressions (Gosman and Pun (1974)).

$$f = \frac{1}{2} \left(1 + \frac{2}{Pe} \right) \quad |Pe| < 2 \quad (4-26)$$

(Derived from central difference expression for total flux)

$$\left. \begin{aligned} f &= 1 & Pe > 2 \\ f &= 0 & Pe < -2 \end{aligned} \right\} \begin{aligned} &\text{Derived from asymptotes} \\ &\text{of the exact solution.} \end{aligned} \quad (4-27)$$

This process is equivalent to employing a central difference finite difference approximation for $|Pe| < 2$ and an upwind finite difference approximation for the convective fluxes when $|Pe| > 2$ and it has therefore, become known as the hybrid difference approximation. It is stable for all grid Peclet numbers.

Where the equation to be solved includes source terms, these are integrated by making the assumptions of linear variation of quantities between nodes and constant generation of the source within the control volume. The resulting integral is expressed in the linearised form :-

$$\int_V S^\phi dV = b\phi_p + c \quad (4-29)$$

where b and c may be functions of ϕ .

Combining equations (4-21) and (4-29) and making use of continuity we obtain the following equation for ϕ_p :

$$(A_p^\phi - b^\phi)\phi_p = A_N^\phi\phi_N + A_S^\phi\phi_S + A_E^\phi\phi_E + A_W^\phi\phi_W + A_R^\phi\phi_R + A_L^\phi\phi_L + C^\phi \quad (4-30)$$

where

$$A_N^\phi = (\rho V_y \Delta x \Delta z)_n (f_n - 1)$$

$$A_S^\phi = (\rho V_y \Delta x \Delta z)_s f_s, \text{ etc}$$

and

$$A_p^\phi = A_N^\phi + A_S^\phi + A_E^\phi + A_W^\phi + A_R^\phi + A_L^\phi$$

In order to make sure that the problem is soluble by ^{the ADI scheme} it is necessary that $(A_p^\phi - b^\phi) > 0$. This is achieved by ensuring that $b^\phi < 0$ in the linearisation of the source term.

A similar process on the appropriate staggered grid control volumes leads to expressions for the momentum equations. The V_x equation is, for example :-

$$(A_p^{V_x} - b^{V_x})(V_x)_p = A_N^{V_x}(V_x)_N + A_S^{V_x}(V_x)_S + A_E^{V_x}(V_x)_E + A_W^{V_x}(V_x)_W + A_R^{V_x}(V_x)_R + A_L^{V_x}(V_x)_L + (\Delta y \Delta z)_l (p_L - p_p) + C^{V_x} \quad (4-31)$$

4.4.2 Calculation of the pressure field

A difficulty of the primitive variable formulation of the Navier-Stokes equations arises from the calculation of pressures.

It is necessary to obtain a pressure field such that the solution of the momentum equations satisfies continuity. This difficulty is overcome by the use of an equation for "pressure correction" derived from a combination of the finite difference equations for momentum and continuity in a guess and correct procedure similar to that of Amsden and Harlow (1970), Chorin (1968) and Patankar and Spalding (1972).

The pressure and velocity fields are initialised and the three momentum equations solved to give a preliminary velocity field which, in general, does not satisfy the continuity equation. Setting ϕ equal to unity the resulting mass imbalance on the control volume is obtained from equation (4-21)

$$m_p^* = -(\rho v_y^* \Delta x \Delta z)_n + (\rho v_y^* \Delta x \Delta z)_s - (\rho v_z^* \Delta x \Delta y)_e + (\rho v_z^* \Delta x \Delta y)_w - (\rho v_x^* \Delta y \Delta z)_r + (\rho v_x^* \Delta y \Delta z)_l \quad (4-32)$$

Assuming that the velocity is a function of pressure only, then we can determine the dependence of the velocity field on the pressure field, i.e.

$$V = V(p) \\ V \simeq V^* + \frac{\partial V^*}{\partial (\Delta p^*)} \cdot (\Delta p^*) \quad (4-33)$$

Where Δp^* is the driving pressure difference.

If the pressure field is corrected by local amounts p' then the above relation gives :-

$$(v_x)_r \simeq (v_x^*)_r + \frac{(\partial v_x^*)_r}{\partial (p_p^* - p_r^*)} \cdot (p_p' - p_r') \quad (4-34)$$

$$(v_x)_l \simeq (v_x^*)_l + \frac{(\partial v_x^*)_l}{\partial (p_l^* - p_p^*)} \cdot (p_l' - p_p')$$

Etc. for v_y and v_z

The coefficients in equation (4-34) are obtained from the finite difference momentum equations (e.g. (4-31)) so that

finally we obtain :-

$$(V_x)_r \simeq (V_x^*)_r + (DV_x)_r (p'_p - p'_r) \quad (4-35)$$

$$(V_x)_l \simeq (V_x^*)_l + (DV_x)_l (p'_l - p'_p)$$

Etc. for V_y and V_z

$$\text{Where } (DV_x)_r = \frac{\delta(V_x^*)_r}{\delta(p'_p - p'_r)} = \left[\frac{\Delta y \Delta z}{(A_p^{V_x} - b^{V_x})} \right]_r \quad \text{Etc.}$$

The unstarred velocities are required to satisfy continuity. Substituting for V_x^* , V_y^* and V_z^* in equation (4-33) using the expressions (4-35) we obtain, after some rearrangement :-

$$A_p^p p'_p = A_N^p p'_N + A_S^p p'_S + A_E^p p'_E + A_W^p p'_W + A_R^p p'_R + A_L^p p'_L + m_p^* \quad (4-36)$$

$$\text{where } A_N^p = (\rho \Delta x \Delta z)_n (DV_y)_n, \quad \text{Etc}$$

$$\text{and } A_p^p = A_N^p + A_S^p + A_E^p + A_W^p + A_R^p + A_L^p$$

Thus the solution of the pressure correction equation provides corrections to the pressure and velocity fields (through equation (4-35)) which drive the solution towards the mutual satisfaction of the momentum and continuity equations.

4.4.3 Solution of the equations

The equations are solved in an iterative manner using an alternating-direction-implicit technique which is a variant of Peaceman and Rachford (1955). The procedure is two step for the two-dimensional calculations and three-step for the three-dimensional calculations. The three step scheme involves the determination of the intermediate solutions,

$$\phi_p^{(1)} = A_N^p \phi_N^{(1)} + A_S^p \phi_S^{(1)} + \left[\sum_{i=E,W,R,L} A_i^p \phi_i^{(0)} + b_p \right] \quad (4-37)$$

Continued

$$\phi_p^{(2)} = A_E^\phi \phi_E^{(2)} + A_W^\phi \phi_W^{(2)} + \left[\sum_{i=N,S,R,L} A_i^\phi \phi_i^{(1)} + b_p \right] \quad (4-37)$$

$$\phi_p^{(3)} = A_R^\phi \phi_R^{(3)} + A_L^\phi \phi_L^{(3)} + \left[\sum_{i=N,S,E,W} A_i^\phi \phi_i^{(2)} + b_p \right] \quad \text{Cont.}$$

At each step of the scheme one co-ordinate direction is treated implicitly and the most recent approximations of ϕ are used in the other directions. The equations (4-37) are solved by an efficient Gaussian elimination method for tridiagonal matrices, sometimes called the Thomas algorithm (Ames (1977)). The three-step sequence may be repeated if necessary, however a single execution is usually adequate.

A single iteration of the numerical method involves the solution of the three momentum equations ((4-31) and counterparts for V_y and V_z), the pressure correction equation (4-36), and equations representing the transport of k , ε and T in the form of (4-30).

The numerical stability and convergence of the solution are aided by the use of under-relaxation when updating the variables on each iteration, i.e.

$$\phi = \lambda (\phi_{\text{new}}) + (1 - \lambda) \phi_{\text{old}} \quad (4-38)$$

A value of $\lambda = 0.5$ is chosen for all variables apart from p' and T for which $\lambda = 1.0$.

Convergence of the solution of each equation is defined by substituting the latest value of each ϕ into the relevant finite difference equation ((4-31), (4-36) or (4-30)) and determining the sum of the absolute value of the residual for each control volume normalised by the maximum value in the field, i.e.

$$\text{Convergence of } \phi \equiv \frac{\sum_{\text{all cells}} |\text{residual } \phi|}{\phi_{\text{max}}}$$

Satisfaction of continuity is defined by the following condition :-

$$\frac{\sum_{\text{all cells}} |m_p^*|}{\text{FLOW IN}} \leq 1 \times 10^{-4} \quad (4-39)$$

where FLOW IN is the mass flux entering the solution domain.

If condition (4-39) is satisfied and the residual sums are small then, provided the solution has ceased to change, it is assumed that a converged solution has been obtained.

4.5 Boundary conditions

To an extent the boundary conditions employed are dependent on the problem under consideration, however, those for the pressure correction equation and for a solid surface are generally applicable. To avoid repetition they will be described here.

4.5.1 Pressure correction equation

Where the velocity component normal to a boundary is specified there is no need to specify the pressure at the boundary. The condition on the pressure correction arises from the requirement that the velocity at the boundary is fixed so that the velocity correction (Equation 4-35) is zero. Hence taking the West boundary for example :-

$$(v_z)_w = (v_z^*)_w \quad ; \quad (Dv_z)_w = 0$$

so that $A_w^p = (\rho \Delta x \Delta y)_w (Dv_z)_w = 0 \quad (4-40)$

Note that when all the boundary conditions are specified in this way the boundary values of pressure do not enter the solution. The pressure field is defined by the specification of the pressure at a single point in the field.

4.5.2 Boundary conditions at a wall

Turbulent flow over a solid surface is characterised by a thin layer of laminar flow immediate to the wall and very rapid variations of the dependent variables over a thin region adjacent to the wall. Although a modified version of the turbulence model (Jones and Launder (1972)) is capable of describing this region, for three dimensional flows the number of mesh points required to resolve this region would be prohibitive. The solution to this problem arises from the experimentally observed fact that the turbulent flow adjacent to a wall is described by a number of universal (or near universal) laws.

The correct conditions to apply at the wall are :-

$$v_x = v_y = v_z = k = \partial \epsilon / \partial n = 0$$

where n is the direction normal to the wall.

The universal laws are employed in the determination of the values at grid nodes adjacent to the wall. For example in the calculation of the velocity components parallel to the wall the wall shear stress is accounted for by specifying a modified diffusion coefficient.

$$\mu_B^* = \tau_B y_p / V_p \quad (4-41)$$

(See figure 4.2 for definition of terms).

For boundary layer flows the turbulent flow near a wall may be described by the logarithmic law of the wall (Schlichting (1968)).

$$V_p = \frac{V_\tau}{K^*} \ln(E y^+) \quad (4-42)$$

$$\text{Where } V_\tau = \sqrt{\frac{\tau_B}{\rho_p}} \quad \text{and} \quad y^+ = \frac{\rho_p V_\tau y_p}{\mu_p}$$

We employ :-

$$K = 0.433$$

$$E = 8.8$$

The turbulence kinetic energy k is approximately proportional to the shear stress at the wall, Tennekes and Lumley (1972) and Launder and Spalding (1972) so that

$$\frac{\tau_B}{\rho k} \simeq \text{constant} = \sqrt{C_\mu} \quad (4-43)$$

Using this (4-42) may be re-written

$$V_p = \frac{\sqrt{\frac{\tau_B}{\rho}}}{K} \ln \left(E \frac{\rho_p y_p C_\mu^{1/4} k_p^{1/2}}{\mu_p} \right)$$

Re-arranging this expression and using (4-43) again we obtain an expression for the modified diffusion coefficient in terms of known quantities.

$$\mu_B^* = \frac{\rho_p K y_p \sqrt{C_\mu^{1/2} k_p}}{\ln \left(E \frac{\rho_p y_p \sqrt{C_\mu^{1/2} k_p}}{\mu_p} \right)} \quad (4-44)$$

For y^+ less than a critical value the laminar form of the wall function is appropriate, Tenneke and Lumley (1972). We use :

$$\mu_B^* = \mu \quad \text{for } y^+ < 10.9$$

In order to specify a boundary condition to apply to the turbulence kinetic energy equation we make use of the fact that in the vicinity of a wall, the generation term reduces to

$$G = \tau \left(\frac{\partial V}{\partial y} \right)_p \quad (4-45)$$

where here V_p refers to the resolved component of velocity parallel to the wall.

Assuming constant shear between the point P and the wall, we may differentiate equation (4-42) to obtain :

$$\begin{aligned} \left(\frac{\partial V}{\partial y} \right)_p &= \sqrt{\tau_B / \rho_p} / \kappa y_p \\ &= \frac{V_p}{y_p \ln(E y^+)} \end{aligned} \quad (4-46)$$

Substituting in (4-45) and making use of (4-43) we obtain

$$G = \rho_p k_p C_\mu^{1/2} \cdot \frac{V_p}{y_p \ln(E y^+)} \quad (4-47)$$

$$\text{where } y^+ = \frac{\rho_p \sqrt{C_\mu^{1/2} k_p}}{\mu_p}$$

This expression is substituted for the generation term in the source expressions for k.

A boundary condition on the dissipation rate of turbulence energy (ϵ) is obtained by making use of the fact that near a wall the turbulence length scale varies linearly with distance from the wall.

Hence :

$$L_e = \frac{k_p^{3/2}}{E_p} = C_\mu^{-3/4} k y_p$$

so that

$$E_p = \frac{C_\mu^{3/4} k_p^{3/2}}{k y_p} \quad (4-48)$$

The value of E is fixed by setting the source terms b^e and c^e to large values (say 10^{30}) so that the remaining terms of the finite difference equation are negligible.

Two different wall boundary conditions are employed for the equation representing the transport of T . For most of the work presented in the following chapters a zero gradient boundary condition has been applied. This is readily achieved by setting the appropriate A_B^T in equation (4-30) to zero. The value of T at the wall is obtained by extrapolation using a parabola with zero gradient at the wall.

For non-adiabatic walls we follow the approach of Launder and Spalding (1974) in which the temperature profile near a wall is related to the velocity profile using :-

$$T^+ = \frac{\rho_p C_p (T_p - T_B) \sqrt{\tau_B}}{-\dot{q}_B'' \rho} = \sigma_e^T \left[\frac{1}{K} \ln(\epsilon y^+) + P(\sigma^T, \sigma_e^T) \right] \quad (4-49)$$

Where $\dot{q}_B''$ is the heat flux to the surface

and

$$P(\sigma^T, \sigma_e^T) = 9. \left[\left(\frac{\sigma^T}{\sigma_e^T} - 1 \right) \left(\frac{\sigma_e^T}{\sigma^T} \right)^{1/4} \right]$$

(see Jayatilaka (1969)).

Setting the appropriate A_b^T to zero again, and replacing $\sqrt{\frac{\tau_b}{\rho}}$ using equation (4-43) we may use equation (4-49) to provide modified source terms b^T and c^T for either the specified wall heat flux or specified wall temperature conditions.

CHAPTER 5

TWO DIMENSIONAL FLUSH SLOT CALCULATIONS

5.1 Introduction

In this chapter a two dimensional form of the method described in chapter 4 is applied to the study of the flow from a two dimensional flush slot into a crossflowing mainstream. This forms a precursor to the main object of the present work, the study of three-dimensional jets in crossflow and film cooling arrangements more representative of current gas turbine cooling configurations.

The flow set up by ejection from a slot into a crossflowing mainstream has been described in chapter 1 (c.f. section 1.3.1). The existence of upstream influences in the pressure field, together with regions of flow reversal necessitate the use of the elliptic flow equations to calculate the flow. In addition, the shear between the jet and mainstream flows results in significant generation of turbulence necessitating the use of turbulence modelling to represent this. The $k-\epsilon$ turbulence model, described in section 4.3.2, is employed for all the calculations presented in this chapter and the justification for this choice is given in section 4.3.1.

Computational details of the two-dimensional slot calculations are presented in the following section. Comparisons between the predictions and measured data are discussed in section 5.3 and concluding remarks are given in section 5.4.

5.2 Computational details

5.2.1 Boundary conditions

The flow is modelled using a rectangular grid which is

arranged to expand away from the critical area of the flow, the downstream edge of the slot. The solution domain and the grid employed are shown schematically in figures 5.1 and 5.2. Since the equations solved are elliptic it is necessary to specify boundary conditions on all the boundaries of the solution domain and these are summarised in table 5.1.

Boundary	V_x	V_y	T	k	ϵ
Upstream	$V_x = V_x(y)$ Eqn. 5-2	$V_y = V_y(y)$ Eqn. 5-3	$T = T_\infty$	$k = k(y)$	$\epsilon = \epsilon(y)$
Downstream	$(V_x)^{N_z} = (V_x)^{N_z-1} + \Delta V_x$ ΔV_x to maintain mass flux balance over domain	$\partial V_y / \partial x = 0$	$\partial T / \partial x = 0$	$\partial k / \partial x = 0$	$\partial \epsilon / \partial x = 0$
Upper	$\partial V_x / \partial y = 0$	$V_y = 0$	$\partial T / \partial y = 0$	$\partial k / \partial y = 0$	$\partial \epsilon / \partial y = 0$
Wall	Eqn. 4-44	$V_y = 0$	$\partial T / \partial y = 0$ or Eqn. 4-49	Eqn. 4-47	Eqn. 4-48
Slot	Eqn. 5-4 to 5-6 or $V_x = V_s \cos \alpha$	Eqn. 5-4 to 5-6 or $V_y = V_s \sin \alpha$	$T = T_s$	$k = k_s$	$\epsilon = \epsilon_s$

Table 5.1 Boundary conditions employed for slot calculations.

Since the velocity component normal to each boundary is specified, as discussed in chapter 4, the appropriate boundary condition on pressure correction is $A_B^P = 0$.

The boundary layer thickness at the upstream boundary is specified by relating the measured data, usually obtained in the

vicinity of the slot, to the thickness at the upstream boundary using the expression for the growth of flat plate, zero pressure gradient turbulent boundary layers given by Schlichting (1968) :

$$\delta = 0.37 x_0 \left(\frac{V_\infty x_0}{\nu} \right)^{-1/5} \quad (5-1)$$

The velocity profile is defined by the $1/4$ th power law approximation

$$\frac{V_x}{V_\infty} = \left(\frac{y}{\delta} \right)^{1/4}, y \leq \delta ; \quad \frac{V_x}{V_\infty} = 1, y > \delta \quad (5-2)$$

Manipulation of equations (5-1) and (5-2) yields the corresponding result for V_y :

$$\frac{V_y}{V_\infty} = 0.029 \left(\frac{y}{\delta} \right)^{3/4} \left(\frac{V_\infty \delta}{\nu} \right)^{-1/4}, y \leq \delta ; \quad \frac{V_y}{V_\infty} = 0.029 \left(\frac{V_\infty \delta}{\nu} \right)^{-1/4}, y > \delta \quad (5-3)$$

The profiles of k and $\mathcal{E}$ at the upstream boundary are derived from the measurements of Klebanoff and Diehl (1951) for turbulent boundary layers on flat plates.

The downstream boundary condition for V_y , T , k , and $\mathcal{E}$ is $\partial \phi / \partial x = 0$. This is also applied to V_x but with the addition of a correction to the velocities, which is independent of y , to maintain the mass flux balance over the solution domain. At the upper boundary a symmetry condition is applied.

The upstream, downstream, and upper boundary conditions are all approximations to the actual behaviour of the flow. Consequently it is necessary to ensure that the boundaries of the solution domain are located in regions of the flow where these are good approximations of the flow behaviour. The arrangements for ensuring the satisfaction of this requirement are described in section 5.2.2.

The interaction between the jet and mainstream flows gives rise to a pressure gradient across the exit of the slot which causes a distortion of the jet velocity profile. No measurements of the jet distortion have been reported for the slot, however the studies of Bergeles (1976), Crabb (1979) and Andreopoulos (1980) include measurements of the distortion in a circular jet issuing into a cross-stream. In addition to these Foss (1980) has completed a detailed flow visualisation study of the flow in the exit region of such a jet. Bergeles (1976) and Andreopoulos (1980) demonstrate that the flow on the downstream side of the hole is accelerated and that on the upstream side retarded in response to the pressure variation across the hole exit; the measurements of Bergeles (1976) are shown in figure 5.3. The extent to which the interaction between the jet and mainstream flows is affected by the shape of the velocity profile is not known, and cannot readily be determined experimentally. Consequently the present solution method has been used to complete a parametric study of the influence of the slot exit boundary condition on the predicted results. The study was carried out for a 90 degree slot, and covered a range of velocity ratios, J , from 0.33 to 1.24. Three forms of slot exit velocity profile were studied :-

- 1) Plug flow, $(V_u)_{slot} = V_T$
- 2) Downstream bias, $(V_u)_{slot} = 2 V_T (x/s - 1)$
- 3) Constant total pressure, Eqns. 5-4 to 5-6.

The influence of the velocity profile form on the solution is summarised in table 5.2.

J	Maximum change in V_x - velocity at $x/D = 2$	Maximum change of η_{ew}
0.33	$0.013 V_\infty$	0.01
0.64	$0.057 V_\infty$	0.05
1.24	$0.186 V_\infty$	0.10

Table 5.2 Maximum difference in the solutions
obtained as a result of changing the slot
exit velocity profile.

It can be seen that for $J = 1.24$ the form of the slot velocity profile has a marked effect on the solution. The influence increases with J because the major effect is on the size of the recirculation region. At low velocity ratios the recirculation is small and consequently exerts little influence on the solution as a whole. Since the recirculation region is larger at higher velocity ratios its influence on the solution increases and with it the effect of the change in the slot velocity profile. Bergeles (1976) has observed that, for jets in crossflow, the hole exit velocity profile is well represented by assuming that the total pressure of the injected flow is constant in the plane of the exit. The velocity profile shape generated by this assumption (form 3, above) lies between that of the plug flow and downstream biased profiles (forms 1 and 2 respectively). At low injection rates ($J \approx 0.3$) the velocity

profile is similar to that of the downstream biased form, whereas at high injection rates ($J \gg 1.0$) the velocity profile is similar to that of the plug flow. Consequently the plug flow velocity profile hole exit boundary condition may be expected to allow accurate predictions for $J \gg 1$ and also for $J \leq 0.3$ for which the influence of the boundary condition on the solution is small. At intermediate velocity ratios, however, it is necessary to employ a boundary condition which provides a more accurate representation of the slot exit velocity profile, such as the constant total pressure form. Since the range of velocity ratios of interest to the present study extends from $J \approx 0.3$ to 2.0 the constant total pressure form of boundary condition has been employed. The assumption of constant total pressure is valid provided the flow path through the slot is relatively short (less than 10 S, say) and that the entry to the slot is well designed thus avoiding flow separation at the inlet. With the additional assumption that the jet fluid exhausts at α , the geometric angle of injection of the slot, the velocity profile may be determined in the following way :-

The area mean total pressure is calculated from

$$\hat{p}_T = \frac{\sum_{m=1}^{I_2} \left[p_m + \frac{1}{2} \rho_m (V_x^2 + V_y^2) \right] \Delta x_m}{\sum_{m=1}^{I_2} \Delta x_m} \quad (5-4)$$

where $\sum_{m=1}^{I_2}$ signifies summation over the finite difference cell faces across the exit of the slot.

The local dynamic head of the injection flow is given by :-

$$Q_m = \hat{p}_T - p_m \quad (5-5)$$

and the jet velocity magnitudes by

$$|V|_m = \sqrt{\frac{2 Q_m}{\rho_m}} \quad (5-6)$$

These values are factored to maintain the correct mass flux through the slot and the x and y velocity components obtained.

This process is applied on each iteration of the solution, allowing the jet velocity profile to develop in response to the computed pressure field. It is possible for Q_m to become negative during the early stages of the convergence of the solution and if this occurs $|V|_m$ is set to a small positive value.

The assumption that the jet fluid exhausts at α across the slot exit is essential to the foregoing analysis, it is not, however, physically reasonable. In chapter 6 the analysis is extended so that the variation of α across the hole exit may be determined. It has also been assumed that k and ϵ are constant in the exit plane of the slot. The parametric study of the slot exit boundary conditions has included an investigation of the importance of the assumed values for k and ϵ , and it has been found that the effect on the solution is not significant.

5.2.2 Determination of the size of the solution domain

As discussed in section 5.2.1 it is necessary to locate the upstream, downstream and upper boundaries of the solution domain in regions of the flow where assumptions made in forming the boundary conditions are satisfied. In particular, in order to minimise the number of grid nodes required for the solution, we seek the smallest solution domain consistent with the above. This is determined by the following computational experiment.

A series of computations are conducted using a coarse finite difference grid in which the location of each boundary is altered by adding additional grid nodes or subtracting grid nodes. The

boundaries of the solution domain are determined by the boundary locations beyond which the solution ceases to change with the addition of further grid nodes. This process has been completed for slots with $\alpha = 30, 60$ and 90 degrees. The results are summarised in table 5.3.

Slot Angle, α (Degrees)	Upstream Boundary (x/S)	Downstream Boundary (x/S)	Upper Boundary (y/S)
30	- 14	18	27
60	- 40	32	46
90	- 75	50	60

Table 5.3 Solution domain boundary locations.

5.2.3 Numerical accuracy

The numerical accuracy of finite difference solutions is dependent on the finite difference approximations employed the distance between the calculation nodes and the nature of the problem for which the solution is sought. It is therefore necessary to determine the extent to which the solution obtained is subject to numerical error. This is normally achieved by completing calculations with reduced spacing between the finite difference grid nodes and inspecting the change in the solution. With the intention of obtaining numerically accurate solutions

this process is repeated until either the limit of the available computer resources is reached, or the solution ceases to change with increasing grid refinement. If grid independent solutions are obtained it is normally assumed that the solution is numerically accurate.

In the present study grid refinement exercises were completed for slots with $\alpha = 30, 60$, and 90 degrees. The influence of the grid refinement on the 60 degree slot solution is shown in figure 5.4 in which the decreasing change in the solution with increasing refinement of the grid may be observed. The finest grid employed represented the maximum that could be accommodated on the available computer resources and it can be seen that a grid independent solution has not been obtained. The final grids resulting from the grid dependence studies are summarised in table 5.4. In each case it has not been possible to obtain grid independent solutions. Typically, the solutions required $300k$ bytes storage, however, it was the CPU time which was limiting factor on the number of grid nodes employed. The solutions for $\alpha = 60$ and 90 degrees required approximately 20 minutes CPU on an IBM 360 . Solutions for the 30 degree slot required over 30 minutes CPU on the IBM 360 and because of this only a limited number of solutions could be obtained.

Angle of Injection (Degrees)	N_x	N_y
30	42	42
60	33	28
90	33	24

Table 5.4 Grids employed for the predictions of section 5.3.

In the initial stages of the grid refinement study the grid line adjacent to the surface was located such that in the near wall region $y^+ > 30$ to ensure the validity of the log-law wall function. For the 30 degree slot however, the solutions failed to resolve the recirculation which has been observed downstream of the slot (Matthews (1972)), and were in poor agreement with measurement. It was clear that the downstream corner of the slot was not being defined accurately. Consequently solutions were obtained with refined grids in the region of the downstream corner of the slot in which the near wall grid line was located at $y^+ \approx 0.3$. The solutions obtained using the refined grids resolved the recirculation and were in much better agreement with measurement. The high Reynolds number form of the $k-\epsilon$ turbulence model employed in the present work is not valid for $\mu_t \sim \mu$, however, and will be inaccurate for $y^+ < 30$. That the refined grid solutions are in good agreement with measured data (c.f. section 5.3.3.) indicates both the strong influence of the pressure field on the flow and the importance of identifying the regions of the solution in which numerical error is significant. Grid dependence tests are a poor measure of numerical accuracy, particularly where the maximum number of grid nodes is limited by computer capacity as was the case for the present solutions.

The hybrid difference scheme (Eqs. 4-26 to 4-28) uses the central difference approximation across cells for $|Pe| < 2$ and upwind differencing for $|Pe| > 2$. In the present calculations $|Pe| > 2$ in at least one co-ordinate direction at almost every grid node. It is therefore appropriate to focus attention on numerical errors arising from the use of the upwind difference approximation in the evaluation of convection terms.

Many studies have been published in which the numerical accuracy of upwind differencing has been examined including Wolfshtein (1968), Raithby (1976a) and McGuirk and Rodi (1978). It has been established that upwind differencing is subject to significant error where high mesh Peclet numbers are associated with regions where the spatial curvature of the solution in the co-ordinate direction is large. An examination of the truncation error of the upwind difference approximation by Taylor series expansions about a grid point indicates that the leading error term has the form of a diffusion flux and as a result the error has become known as numerical diffusion. De Vahl Davis and Mallinson (1972) use the truncation series to derive an expression for the coefficient of numerical diffusion in terms of the local grid spacings and velocity components. In isolation the coefficient of numerical diffusion is not particularly useful since it is only a measure of numerical error in combination with the second derivative of the dependent variable. In addition it is misleading to use the relative magnitudes of the numerical and physical diffusion terms as a measure of the numerical accuracy of a solution since it is only when the numerical diffusion error becomes significant with respect to all the remaining terms in the discretised transport equation that large errors will arise. McGuirk and Rodi (1978) describe a method for estimating the local numerical diffusion of solutions obtained using upwind differences. An estimation of the local truncation error is obtained from the converged solution of the finite difference equations by evaluating the convection, diffusion and source and sink terms using central difference approximations and determining the out of balance residual at each grid node. The out of balance residual is the difference in the truncation series for upwind and central

differences and the leading term is the numerical diffusion term. The importance of numerical error in the solution is determined by comparing the local magnitude of the out of balance residual with that of the convection, diffusion and source and sink terms of the transport equation. McGuirk, Taylor and Whitelaw (1981) employed the method of McGuirk and Rodi (1978) to guide the grid refinement of the solution for the flow downstream of an axisymmetric baffle. By concentrating grid refinement in regions where the out of balance residual constituted an important term in the local balance of the transport equation, a solution was obtained in which the numerical error was reduced to insignificant levels. It was observed that numerical diffusion was worst within the first quarter of the recirculation and that the length of the recirculation increased as the grid was refined.

A comparison between measurements of the length of the recirculation downstream of angled flush slots and the predictions of the present study is shown in figure 5.5. It can be seen that the results for the 30 and 60 degree slots indicate a slight tendency to underpredict the measurements but that, for jet to mainstream momentum ratios greater than unity, the recirculation length downstream of 90 degree slots is seriously underpredicted (by up to 45%). In view of the observation of McGuirk, Taylor and Whitelaw (1981) this underprediction suggests that the solutions for $J > 1$ on the 90 degree slot configuration are most prone to numerical diffusion errors. One of these cases has been chosen for examination by the method of McGuirk and Rodi and the results for the axial-momentum equation and the conserved scalar are shown in figures 5.6(a) and (b) respectively. It can be seen that the largest errors arise in the shear layers between the jet and the mainstream and between the jet and the recirculation. It

is in these regions that large second derivatives arise in the co-ordinate directions in combination with large grid Peclet numbers and consequently the numerical diffusion error becomes significant. These errors will result in the diffusion of momentum from the jet so that the peak velocity of the jet will tend to be underpredicted. Numerical diffusion within the recirculation results in the underprediction of the reattachment length and errors in the adiabatic wall effectiveness predictions. From figure 5.6(a) it can be seen that the axial-momentum equation is dominated by the influence of pressure gradients so that the effect of numerical diffusion errors is reduced. This is not the case for the conserved scalar, figure 5.6(b), in which the transport equation represents a balance between the convective and diffusive terms. Consequently the numerical diffusion error is a more important term in the local balance of the discretised transport equation for conserved scalars. Since in the present work the dependance of the momentum equation on the conserved scalar is weak, then the numerical accuracy of the velocity field predictions will be better than that of the conserved scalar.

The numerical diffusion errors of a 60 degree slot solution have also been examined using the technique of McGuirk and Rodi and in general the results were similar to those obtained for the 90 degree slot, showing a slight reduction in the relative magnitude of the error terms. In the region of the slot exit however, where steep gradients in the dependant variables are convected at an angle to the mesh, the contribution of the error term was much greater than in the solution for the 90 degree slot. It is because of the large errors arising in the immediate vicinity of the slot exit that the fine grid was required for the 30 degree slot configuration.

The estimations of numerical diffusion presented indicate that in small regions of the flow the numerical error becomes the dominant term in the equations solved. The estimated error is related to the error in determining the gradients of the dependent variables and that these are not necessarily reflected in large errors in the predicted values of the dependent variables. Castro (1979) has observed that the existence of local regions of significant numerical diffusion error appeared to have little effect on the accuracy of the solutions obtained.

5.2.4 Turbulence modelling

The $k-\epsilon$ turbulence model employed for the present predictions has been applied to a wide range of two dimensional shear flows with a considerable degree of success. It is because of this, and also as a result of its modest computational requirements that the $k-\epsilon$ model has been chosen for use in the present work (c.f. section 4.3.1). As observed by Rodi (1981), however, turbulence models based on an isotropic effective viscosity (Eqn. 4-16) have shortcomings when used to model complex flows, for which the effective viscosity and diffusivity cannot be assumed to be isotropic but should depend on the stress or flux component considered. Further, it is assumed that the local state of turbulence can be characterised by one velocity scale $\sqrt{k}$ to which the individual Reynolds stresses can be related but, in reality, the individual Reynolds stresses may develop quite differently in the flow. One of the consequences of these shortcomings is that the normal stresses are not accurately represented. This leads to inaccuracies in the computation of the recirculation since, as

observed by McGuirk, Taylor and Whitelaw (1981), turbulence production through normal stress normal strain is important within regions of recirculation.

In a study of two-dimensional reattaching jet flows Sawyer (1963) has observed that streamline curvature had a marked effect on the entrainment properties of the jets. Bradshaw (1973) has observed that streamline curvature produces appreciable changes in the structure of turbulent stresses and that flows in which $\delta(r-v_\theta)/\delta r$ is positive display a suppression of turbulent mixing and those in which $\delta(r-v_\theta)/\delta r$ is negative give enhanced turbulent mixing. Using this as a criteria, the regions of enhancement and suppression of turbulent mixing in the flow from a flush slot into a crossflow have been determined, figure 5.7. It can be seen that the mixing is enhanced in the shear layer between the jet and the mainstream flow but, is suppressed in the shear layer between the jet and the recirculation. Small regions of enhanced turbulent mixing also arise near the wall in the region of the slot exit and the reattachment point.

Downstream of the recirculation the flow becomes characteristic of far wakes for which dissipation and convection are the dominant terms. Launder et al (1972) have shown that in these circumstances C_μ (Eqn. 4-15) can no longer be considered constant but becomes dependent on the local ratio of production to dissipation.

Finally, as discussed in the previous section, the high Reynolds number form of the $k-\epsilon$ model is inappropriate for the solutions obtained of the 30 degree slot configuration. The low Reynolds number form of the $k-\epsilon$ model of Jones and Launder (1972) would be more appropriate, however, relative to the high Reynolds

number form of the model this requires a considerable increase in the number of cross-stream grid nodes to ensure numerical accuracy and this would have exceeded the capacity of the computer resources available.

Many of the shortcomings of the $k-\epsilon$ turbulence model described here can only be overcome by the use of a turbulence model in which equations representing the transport of the individual Reynolds stresses and turbulent heat fluxes are solved. As discussed in section 4.3.1 this necessitates an increase in the computer resources required for the solution, however, as shown in the previous section, the computer resources available for the present solutions have not been sufficient to allow the eradication of numerical error. In these circumstances there is no justification for the adoption of a more sophisticated turbulence model.

5.3 Comparison between prediction and measurement

The accuracy of the present solutions has been studied by comparing the predictions with the experimental data of Artt (1970), Foster (1973) and Sivasegaram (1967) for injection angles, α , of 30, 60 and 90 degrees.

5.3.1 Ninety degree slot results

Comparisons with the velocity profile measurements of Foster (1973) are presented in figure 5.8. For $J = 0.64$ and 0.97 the predictions are in excellent agreement with the measurements, however, for $J = 1.24$, the extent of the recirculation has been underpredicted by at least 20%. As discussed in section 5.2.3 this is largely a result of the numerical diffusion errors arising from the use of upwind differences to approximate the convection

terms of the equations. It has been shown in section 5.2.3 that the shear layers on either side of the jet are most prone to numerical diffusion. As J increases, the size of the recirculation increases so that the shear layers stray into coarser parts of the finite difference mesh leading to an escalation of the numerical error at the higher values of J .

Figure 5.9 includes comparisons between the predictions and the adiabatic wall effectiveness measurements of Foster (1973) and Sivasegaram (1967). The low velocity ratio conditions have been predicted to within the experimental error, however for $J > 1$ there is a tendency to overpredict η_{aw} . This arises from a combination of three effects. Firstly, there will be a tendency for η_{aw} to be overpredicted as a result of the underprediction of the length of the recirculation downstream of the slot. Secondly the numerical diffusion errors arising in the solution of the equation for the conserved scalar result in a diffusion of the jet fluid into the mainstream and into the recirculation over and above that arising from physical sources. To an extent the error arising from the diffusion of jet fluid into the mainstream as a result of numerical diffusion is offset by the fact that the isotropic eddy viscosity formulation of the $k-\epsilon$ turbulence model will result in the underprediction of the turbulent mixing between the jet and the mainstream due to the streamline curvature. This is not the case in the shear layer between the jet and the recirculation so that the diffusion of the jet fluid into the recirculation is overpredicted as a result of both numerical error and shortcomings of the turbulence model. Finally, the rate of decrease of η_{aw} with increasing downstream distance tends to be underpredicted beyond $x/S = 20$. The flow in this region is characteristic of far wakes and consequently, as

discussed in section 5.2.4, C_μ requires modification. The increase in μ_e which would result would give rise to an increase in the rate of decrease of η_{ew} with downstream distance, tending to improve the agreement with the measured results.

The heat transfer measurements of Foster (1973) were arranged so that the temperatures of the mainstream and injected fluids were identical, removing the difficulty of defining a "driving temperature" for the heat transfer. Comparisons of the predictions with these measurements are presented in figure 5.10. At all three conditions studied the Stanton number is underpredicted (by up to 25% for $J = 0.59$ increasing to 40% for $J = 1.14$). The underprediction arises from several causes. As a result of the incorrect calculation of the normal stresses within recirculation regions (c.f. section 5.2.4) the $k-\epsilon$ turbulence model has been found to underpredict the levels of k within recirculations, McGuirk, Taylor and Whitelaw (1981), and this will result in the underprediction of the wall heat flux. Additionally it can be seen from figure 5.7 that streamline curvature will result in regions of enhanced turbulent mixing near the wall. Once again the turbulent kinetic energy, k , will be underpredicted in these regions and with it the wall heat flux. A further cause of the underprediction arises from the form of the wall functions used (equations 4-42 to 4-49). Since this work was completed Chieng and Launder (1980) have derived new wall functions which have given improved predictions of wall heat flux for flows with recirculation.

5.3.2 Sixty degree slot results

Comparisons between the predicted velocity profiles and the measurements of Artt (1970) are shown in figure 5.11. It can be seen that, for $J = 1.25$, the agreement between prediction and

measurement is generally within about 10% except for the peak velocity at $x/S = 1.0$ which is underpredicted by about 20% largely as a result of numerical diffusion. For $J = 0.75$ however, significant discrepancies exist between the measurements and the predicted results. The measurements suggest that the flow reattaches to the surface in the vicinity of $x/S = 2.0$. This does not agree with the measurements of Wygnanski and Newman (1968) which indicate a reattachment point at $x/S = 5.5$, in good agreement with the predicted result ($x/S = 5.3$). In his experiment Artt observed that the flow within the recirculation was three-dimensional and it would appear that this has influenced the measurements for this condition.

Comparisons with the adiabatic wall effectiveness data of Sivasegaram (1967) are shown in figure 5.12. The agreement between the measured and predicted results is similar to that observed previously for the ninety degree slot however, the overprediction of η_{aw} for $J = 1.45$ is somewhat worse for the sixty degree slot configuration. This arises because the numerical diffusion error in the immediate vicinity of the slot exit is greater for the 60 degree slot solutions (c.f. section 5.2.3).

5.3.3 Thirty degree slot results

The predictions of the velocity profiles downstream of a thirty degree slot are in fair agreement with the measurements of Artt (1970), figure 5.13. The maximum discrepancy is associated with the underprediction of the velocity peak at $x/S = 1.0$ and as with the 60 degree slot this is caused by numerical diffusion. Comparisons with the adiabatic wall effectiveness measurements of Sivasegaram (1967) are shown in figure 5.14. Agreement to within the experimental error has been obtained.

5.4 Concluding remarks

The flowfield set up by ejection from a two-dimensional slot into a crossflowing mainstream is strongly influenced by the velocity profile of the fluid exhausting from the slot for $J > 0.6$. The simple constant total pressure boundary condition employed in the present work allows the slot velocity profile to develop in response to the pressure field generated by the interaction between the jet and mainstream flows. Provided errors from other sources are not significant, predictions employing this boundary condition are in good agreement with experimental measurements.

An examination of the numerical diffusion error of the solutions arising from the use of the upwind difference approximation has shown that for $J > 1$ significant numerical diffusion exists in some regions of the solution. The numerical error is particularly significant in the shear layers between the jet and the mainstream and between the jet and the recirculation. For $J < 1$ errors arising from numerical diffusion do not appear to be significant and the predictions are in good agreement with experimental measurements. This suggests that the shortcomings in the $k-\epsilon$ turbulence model discussed in section 5.2.4 are not significant at the lower velocity ratios. For $J > 1$ numerical diffusion errors become significant in the region of the recirculation so that the performance of the $k-\epsilon$ turbulence model cannot be criticized for this part of the flow. Beyond $x/S = 20$ however, numerical error is not significant so that we may tentatively ascribe errors in this region to the turbulence modelling. It is possible that the modified form of $k-\epsilon$ model described by Launder et al (1972) would result in an improvement of the agreement with

experimental measurement.

The present solutions for the 30 degree slot have shown considerable sensitivity to grid refinement as a result of the large numerical errors that arise in the vicinity of the slot exit with a coarse solution mesh. In view of the small size of the recirculation region it is probable that better results would be obtained with a suitably modified partially parabolic procedure, since this computationally efficient method would allow finer solution meshes.

CHAPTER 6

JET IN CROSSFLOW CALCULATIONS

6.1 Introduction

In this chapter the three dimensional procedure of Jones (1976) is applied to the solution of the flow set up by a jet exhausting perpendicularly from a flat surface into a crossflowing mainstream. The ejection of a secondary fluid in this manner causes a pressure difference to be set up across the hole exit which results in the distortion of the flow within the injection hole (c.f. section 5.2.1). The two-dimensional calculations of chapter 5 demonstrated the importance of correctly defining the velocity profile of the jet exit and a simple boundary condition based on an observation of Bergeles (1976) was employed to represent this. A limitation of this boundary condition is that the cross-flow components set up within the hole as a result of the distortion are neglected. The measurements of Andreopoulos (1980) have shown, however, that these cross flow velocity components can be significant in relation to the nominal jet velocity, V_j .

A novel feature of the work presented in this chapter is the use of the numerical procedure to compute the distortion of the flow within the ejection hole. This removes the limitation of the approach adopted in chapter 5 since the solution provides the full three-dimensional velocity field within the hole and the crossflow velocity components at the hole exit are determined. Solutions for the complete flowfield are therefore obtained from two computations : one, using a rectangular mesh, solving for the jet-mainstream interaction and the other, using a cylindrical grid, for the flow within the hole. Iteration is carried out between

the two computations to obtain converged solutions of the flow-field. In a recent publication Kreskovsky Briley and McDonald (1980) present calculations in which a single computation has been employed to solve for both the jet-mainstream interaction and the distortion of the flow within the hole. The calculation employs a cylindrical mesh centred on the hole axis. This is not suited to the computation of the mainstream flow since there are large regions within which the flow is not aligned with the mesh so that the solution will be prone to numerical error (c.f. section 5.2.3). The rectangular mesh employed in the present work is better suited to the solution of the jet-mainstream interaction since the flow is more closely aligned with the grid so that the numerical accuracy of the solution is potentially better than that of Kreskovsky et al (1980). The present calculations also make more efficient use of the available mesh nodes, since the solution meshes may be individually tailored to suit the flow within each calculation domain.

The computational details of the solution for the jet-mainstream interaction are described in the following section. An assessment of the need for computation of the crossflow velocity components within the hole is presented in section 6.3, which also includes a description of the computational details of the solution within the hole and the arrangements for linking the two solutions. Comparisons between the predicted results and experimental measurement are presented and discussed in section 6.4 and concluding remarks are given in section 6.5.

6.2 Computational details of the solution for the jet-mainstream interaction

6.2.1 Boundary Definition

The jet mainstream interaction is modelled on a rectangular grid within the domain shown in figure 6.1. Boundary conditions are required on all six boundaries of the domain and these are summarised in table 6.1. The upstream boundary conditions are similar to those employed in the two dimensional computations presented in chapter 5 (c.f. section 5.2.1) except that the profiles of k and ϵ are derived from the mixing length expression for two dimensional boundary layers, using :

$$k = \frac{\tau}{\rho \sqrt{c_\mu}} = \frac{l_m^2}{\sqrt{c_\mu}} \left(\frac{\partial V_x}{\partial y} \right)^2 \quad (6-1)$$

$$\epsilon = \frac{k^{3/2}}{L} = \frac{(c_\mu^{1/2} k)^{3/2}}{l_m} \quad (6-2)$$

Where l_m is Prandtl's mixing length :-

$$\begin{aligned} l_m &= \kappa y \quad \text{for } y \leq c_\mu \delta / \kappa \\ &= c_\mu \delta \quad \text{for } y > c_\mu \delta / \kappa \end{aligned}$$

Further details of this treatment may be found in Launder and Spalding (1974).

Boundary	V_x	V_y	V_z	T	k	ϵ
Wall	Eqn. 4-44	$V_y = 0$	Eqn. 4-44	$\partial T / \partial y = 0$ or $\dot{q}$ from Eqn. 4-49	Eqn. 4-47	Eqn. 4-48
Hole	See sections 6.3.2 and 6.3.4					
Top	$\partial V_x / \partial y = 0$	$V_y = 0$	$\partial V_z / \partial y = 0$	$\partial T / \partial y = 0$	$\partial k / \partial y = 0$	$\partial \epsilon / \partial y = 0$
Lateral : Symmetry through hole centre	$\partial V_x / \partial z = 0$	$\partial V_y / \partial z = 0$	$V_z = 0$	$\partial T / \partial z = 0$	$\partial k / \partial z = 0$	$\partial \epsilon / \partial z = 0$
Lateral : Far Boundary	$\partial V_x / \partial z = 0$	$\partial V_y / \partial z = 0$	$\partial V_z / \partial z = 0$	$\partial T / \partial z = 0$	$\partial k / \partial z = 0$	$\partial \epsilon / \partial z = 0$
Upstream	Eqn. 5-2	Eqn. 5-3	$V_z = 0$	$T = T_\infty$	Eqn. 6-1	Eqn. 6-2
Downstream	$\partial V_x / \partial x = 0$ + ΔV_x to maintain mass flux balance over domain	$\partial V_y / \partial x = 0$	$\partial V_z / \partial x = 0$	$\partial T / \partial x = 0$	$\partial k / \partial x = 0$	$\partial \epsilon / \partial x = 0$

Table 6.1 Boundary conditions employed for the computation of the jet-mainstream interaction.

The definition of a circular hole by the rectangular grid poses problems and it is necessary to ensure that the fluxes of quantities entering the solution domain match those leaving the hole. This is achieved by modifying the surface area of boundary cell faces at the edge of the hole so that the correct fluxes of mass and momentum from the hole are conserved. Consideration is given to suitable forms of hole exit boundary condition in sections 6.3.2 and 6.3.4.

The size of the solution domain has been minimised by the use of the computational experiment technique described in section 5.2.2. It is appropriate to scale the size of the solution domain according to the penetration and lateral spread of the injected flow. Accordingly the computational experiment has been completed for two cases corresponding to the maximum and minimum injection rates included in the present study. The size of solution domain required for a particular injection rate may be determined by interpolation between these cases on the basis of the jet-mainstream momentum ratio. The boundary locations determined by the computational experiment are shown in table 6.2.

Boundary		Momentum Ratio = 0.058	Momentum Ratio = 4.0
Upstream	x/D	- 2.0	- 2.0
Downstream	x/D	5.5	5.5
Lateral	z/D	2.0	3.0
Top	y/D	4.0	6.0

Table 6.2 Boundary locations determined for the solution of the jet-mainstream interaction.

6.2.2 Numerical Accuracy

An initial estimate of the numerical accuracy of the solutions has been obtained by establishing the grid dependence of the solution by completing a series of calculations on successively refined finite difference meshes. The effect of the grid refinement on the V_x -velocity profile downstream of the hole is shown in figure 6.2. It can be seen that as the grid is refined the velocity profile starts to develop a peak. Corresponding changes are also seen in the V_y - velocity profiles. The finest grid employed of $25 \times 20 \times 15$ nodes ($N_x \times N_y \times N_z$) still shows evidence of grid dependence however, this represented the practical limit of refinement on the available computer facilities. Although grid independent solutions have not been obtained, the predictions are not grossly in error. For example, in comparison with the measurements of Ramsey and Goldstein (1971), the solution obtained on the finest grid under-predicts the peak V_x - velocity by about 17%. The $25 \times 20 \times 15$ node grid has been employed for all subsequent predictions of the jet-mainstream interaction presented in this chapter.

A more detailed examination of the numerical accuracy of the solutions has been completed using the method of McGuirk and Rodi (1978) described in section 5.2.3. This method of estimating the truncation error of the upwind difference approximation is appropriate since upwind differencing is employed in at least one co-ordinate direction throughout the entire solution domain. Profiles of the various terms and the out of balance residual for the x - direction momentum and conserved scalar equations are shown in figure 6.3 (a) and (b) respectively. The results for the momentum equation show that at $y/D = 0.23$ the numerical

diffusion error is generally small but that some errors arise at the jet boundaries. Further from the wall at $y/D = 0.60$ significant errors arise in the reverse flow region but elsewhere the error is not significant. The greatest numerical diffusion errors occur at $y/D = 1.05$ where steep gradients in the jet are convected at an angle to the grid, which in turn becomes progressively coarser with increasing distance from the wall. The numerical diffusion errors decrease beyond $y/D = 1.05$ as the flow deviations from the co-ordinate directions and the gradients in the V_x -velocity become smaller.

The relative magnitudes of the numerical diffusion errors for the conserved scalar equation are greater than those for the x - momentum equation. The numerical diffusion is significant throughout most of the jet up to $x/D = 2.0$.

6.2.3 Continuation downstream

The extent of the region downstream of a row of film cooling holes which is of interest to turbine blade cooling design may extend beyond 100 x/D 's. The domain employed for the computation of the jet-mainstream interaction extends only to $x/D = 5.5$ so that it is necessary to continue the solution further downstream.

Ideally the method employed for the continuation should be rapid, have low storage requirements and retain the three-dimensional character of the solution. Marching methods based on the parabolised Navier-Stokes equations (e.g. Patankar and Spalding (1972))

fulfil the above requirements, however, terms removed in the process of parabolising the equations limit their range of application. In particular, it is necessary that the streamwise pressure gradient be constant on cross-stream planes and equal to the free-stream value.

The computational experiment described in section 6.2.1 fixes the location of downstream boundary of the solution at the point where the zero-pressure gradient boundary condition produces a negligible upstream influence in the pressure field. Unfortunately the more restrictive requirement on the validity of parabolic solution methods is not satisfied at this location and it has been necessary to continue the solution to a location beyond which parabolic methods are valid ($x/D \sim 10$) by a further application of the elliptic method. The parabolic solution method employed for computation further downstream is based on the method of Patankar and Spalding (1972).

The lateral, wall, and top boundary conditions employed for the downstream continuation are the same as those employed for the jet mainstream interaction, summarised in table 6.1. In order to retain the jet within the solution domain, the vertical and lateral extents of the solutions have been increased by 1 D over those employed for the jet-mainstream interaction. The meshes for the downstream computations are based on that used for the downstream region of the jet-mainstream interaction.

6.3 Computational details of the solution within the hole

6.3.1 The need for computation within the hole

In order to assess the influence of the jet velocity profile and cross-stream velocity components on the jet-mainstream interaction, solutions have been obtained for a range of injection

rates ($J = 0.5, 1.0$ and 2.0) with several alternative forms of hole exit boundary condition. The results at all three velocity ratios were found to be significantly affected by the hole exit boundary condition, however, the influence tended to be greater at the higher velocity ratios, the maximum effect occurring for $J = 1.0$. The effect on the V_x - velocity profiles for $J = 1.0$ is shown in figure 6.4. It can be seen that changing the form of the jet velocity profile alone may have a significant effect on the solution, the results for the plug flow and the constant total pressure treatment based on Eqns. 5.4 to 5.6 show discrepancies of up to 25% of V_∞ . The results in which the distortion of the velocity profile and the cross-stream velocity components have been determined by computing the flow within the hole are in significantly better agreement with the measured data and demonstrate the need to account fully for the development of the flow within the ejection hole. In the present work the flowfield within the hole is determined by the three-dimensional procedure of Jones (1976). The details of this solution and the arrangements for linking it with that of the jet mainstream interaction are described in the following sections.

6.3.2 Boundary definition

The computation within the hole is modelled on a cylindrical grid within the domain shown in figure 6.5. Boundary conditions are required on all six boundaries of the domain and these are summarised in table 6.3. Two alternative forms of exit boundary condition have been employed : Fixed pressure and fixed V_y -velocity. The reasons for these alternative treatments and the final form used in the predictions are discussed in section 6.3.4.

It is necessary to fix the location of the inlet boundary by computational experiment. The greatest axial extent required for

Boundary	V_a	V_β	V_γ	k	ϵ	p'	p
Wall	$V_a = 0$	Eqn. 4-44	Eqn. 4-44	Eqn. 4-47	Eqn. 4-48	-----	-----
Hole Axis	* $V_a = V_{ax} \cos \beta$	* $V_\beta = -V_{ax} \sin \beta$	----- Mean of adjacent values -----			-----	-----
Symmetry plane through hole centre $\beta = 0, \pi/2$	$\partial V_a / \partial \beta = 0$	$V_\beta = 0$	$\partial V_\gamma / \partial \beta = 0$	$\partial k / \partial \beta = 0$	$\partial \epsilon / \partial \beta = 0$	-----	-----
Inlet	$V_a = 0$	$V_\beta = 0$	$V_\gamma = V_j$ or Eqn. 5-2 with $\delta = D/2$ and $V_{ax} = V_j / 1.817$	$k = k_j$ or Eqn. 6-1 with $\delta = D/2$	$\epsilon = \epsilon_j$ or Eqn. 6-2 with $\delta = D/2$	-----	-----
Exit (1) Fixed Pressure	Linear Extrapolation		Adjusted to maintain mass balance on exit boundary cells	Linear Extrapolation		$p' = 0$	Interpolated from solution of jet-mainstream interaction
Exit (2) Fixed V_γ Velocity	Linear Extrapolation		Interpolated from solution of jet-mainstream interaction	Linear Extrapolation		-----	-----

$$* V_{ax}(n_\gamma) = \frac{1}{2} [V_a(3, 1, n_\gamma) - V_a(3, N_\beta, n_\gamma)] \quad \text{where } V_a(n_\alpha, n_\beta, n_\gamma).$$

Table 6.3 Boundary conditions for the solution within the hole.

the solution will occur at low injection rates, for which the distortion of the jet is most marked. Consequently the computational experiments were carried out for a "worst case" condition using the exit velocity profile for $J = 0.1$ measured by Bergeles (1976). At this condition the distortion was found to extend $1.5 D$ into the hole and this has been used as the inlet boundary location for all subsequent calculations.

6.3.3 Numerical accuracy

The numerical accuracy of the calculations has been tested by investigating the grid dependence of the solution. Calculations were completed using grids of $6 \times 9 \times 8$, $8 \times 9 \times 12$ and $13 \times 13 \times 16$ nodes ($N_a \times N_b \times N_z$). The results for the two finer grids were virtually identical. Further confirmation of the numerical accuracy of the solutions has been sought by comparing predictions obtained on the $8 \times 9 \times 12$ grid with the measurements of Andreopoulos (1980) and, as shown in figure 6.6, agreement to within 7% of V_j has been obtained. The predicted results employed in this comparison were obtained from a converged solution of a linked calculation (hole + jet-mainstream interaction) so that this demonstrates the adequacy of the arrangements for linking the two solutions as well as the numerical accuracy of the computation within the hole. The $8 \times 9 \times 12$ grid has been employed for all the remaining work presented in this chapter.

6.3.4 Linkage with the solution of the jet-mainstream interaction

Different arrangements for linking the solution within the hole with that of the jet-mainstream interaction are required depending on the form of the hole exit boundary condition and the arrangements are summarised in table 6.4.

Step	Operation (Fixed pressure)	Operation (Fixed velocity)
1	Run the computation of the jet main-stream interaction for, say, 60 iterations, using a plug flow boundary condition on V_3 at the hole exit.	Run the computation of the jet-mainstream interaction for, say, 60 iterations, using the 3-D analogue of Eqns. 5-4 to 5-6 to determine V_3 at the hole exit on each iteration.
2	Interpolate the pressure field generated across the hole exit onto the hole solution grid.	Interpolate the V_3 - velocity distribution across the hole exit onto the hole solution grid.
3	Obtain a converged solution of the flow within the hole using the fixed pressure exit boundary condition.	Obtain a converged solution of the flow within the hole using the fixed velocity boundary condition.
4	Transform velocity components at the hole exit from cylindrical co-ordinates to rectangular co-ordinates and interpolate the distributions of V_x , V_y , V_z , κ and ϵ at the hole exit onto the solution grid for the jet- mainstream interaction.	
5	Continue the solution for the jet-mainstream interaction for a further 30 iterations.	Continue the solution for the jet-mainstream interaction for a further 30 iterations using the 3-D analogue of Eqns. 5-4 to 5-6 to update the V_3 - distribution at the hole exit on each iteration.
6	Test for convergence If unconverged : repeat from step 2.	

Table 6.4 Link arrangements for combined solutions of the jet-mainstream.

In both the fixed pressure and fixed velocity formulations of the hole exit boundary condition the assumption has been made that the shear and normal stresses in the plane of the hole exit are not important and that the flow is pressure dominated. The comparison with the measurements of Andreopoulos (1980) shown in figure 6.6 indicates that this assumption is justified. The fixed velocity form of boundary condition involves the additional assumption that the total pressure of the secondary flow is constant in the plane of the hole exit. In order to assess the influence of this assumption on the solution of the jet-mainstream interaction calculations were completed with the following three sets of boundary conditions :-

- i) Inlet : Fully developed turbulent pipeflow
Exit : Fixed pressure form
- ii) Inlet : Plug flow
Exit : Fixed pressure form
- iii) Inlet : Plug flow
Exit : Fixed velocity form

Cases (ii) and (iii) are virtually equivalent and resulted in solutions that were almost identical. The influence of the cases (i) and (iii) on the solution is shown in figure 6.4 and again the results can be seen to be in close agreement. These results suggest that the hole inlet boundary condition has little influence on the jet-mainstream interaction, a conclusion which receives confirmation from the experimental measurements of Goldstein, Eckert and Burggraff (1974) who found η_{aw} was unaffected by the length - diameter ratio of the secondary injection tubes. Although the results obtained using the alternative forms of hole exit boundary condition are in close agreement, the rate

of convergence of the solutions are quite different. The fixed velocity form of exit boundary condition gave much more rapid convergence, particularly at low injection rates ($J < 1$) and consequently this has been employed for the predictions presented in section 6.4.

Convergence of the full solution is assumed to be obtained when both solutions satisfy the mass flux criterion Eqn. (4-39) and the solution of the jet-mainstream interaction ceases to change on the recalculation of the flow within the hole. Using the fixed velocity form of hole exit boundary condition three cycles of the calculation sequence (steps 2 to 6 in table 6.4) are generally sufficient to obtain a converged solution although more cycles may be required at low blowing rates ($J < 1$). A full solution with three cycles, completed on an IBM 370-168 requires a maximum storage of 800k bytes and 50 minutes CPU. The elliptic calculation used to extend the solution downstream requires 400 k bytes and 4 minutes CPU and the parabolic calculation 220 k and 1 minute CPU per 10 x/D 's of downstream distance.

6.4 Comparison with experimental measurements

6.4.1 Velocity profiles

The Laser Doppler anemometry measurements of Crabb (1979) provide further confirmation of the adequacy of the present linked solution arrangements. A comparison between the measured jet velocity profile for $J = 2.3$ at $y/D = 0.25$ and the present predictions is shown in figure 6.7 and it can be seen that excellent agreement has been obtained. Also shown in the figure is the result of a prediction obtained by Crabb using a similar numerical method but with a hole exit boundary condition based on fully - developed turbulent pipeflow profiles. The level of agreement

between measurement and prediction is significantly better with the present prediction than that of Crabb.

Further comparisons with the velocity measurements of Crabb (1979) are presented in figure 6.8. In general, the present predictions are in good agreement with the V_x - velocity profiles shown in figure 6.8 (a) however, a flow reversal is predicted at the downstream edge of the hole which is not shown in the measurements and the peak in the velocity profiles downstream of the hole is underpredicted. The V_y - velocity profiles shown in figure 6.8 (b) are also in good agreement with the measurements, however, the predictions tend to overpredict the velocity peak near the wall and underpredict the velocity peak far from the wall. In the immediate vicinity of the hole the observed discrepancies may be due to the manner in which the hole is represented by the rectangular grid. Further downstream the discrepancies in the V_x - velocity profile are largely caused by numerical diffusion errors : as shown in figure 6.2, the peak in the V_x - velocity profile begins to develop as the grid is refined.

The velocity profile measurements of Ramsey and Goldstein were completed for velocity ratios, J , of 2 and 1 and comparisons of the predictions with this data are shown in figure 6.9. As with the previous results the predictions show a tendency to underpredict the peak velocity but are otherwise in good agreement with the measurements. It can be seen however, that beyond $x/D = 5$ the recovery of the velocity profile towards boundary layer form is somewhat too slow. This region of the flow is wake-like in character and the turbulence energy dissipation rate ($\mathcal{E}$) is much greater than the rate of production ($\mathcal{G}$). As observed in section 5.2.4 under these circumstances the value of C_μ increases

leading to an increase in μ_t . The slow recovery of the velocity profiles downstream of $x/D = 5$ may therefore be caused by the assumption of a constant C_μ in the $k - \epsilon$ turbulence modelling, however, this conclusion must be viewed with caution because of the small magnitude of the effect and because of the existence of numerical diffusion errors in the solutions.

The velocity profile measurements of Bergeles (1976) cover the range $0.24 \leq J \leq 1.0$ and comparisons with these measurements are shown in figure 6.10. Again, the predictions generally agree with the measurements to within the experimental error. Further support is also given to the conclusion that μ_t is underpredicted in the wake-like region in the lee of the jet.

6.4.2 Effectiveness contours

Comparisons with the non-dimensional temperature (effectiveness) contour plots of Ramsey and Goldstein (1971) are presented in figure 6.11. Apart from the region in the vicinity of the wall, the jet spread and penetration are well predicted. It can be seen however that there is a tendency for the peak effectiveness to be underpredicted. In common with the V_x - velocity profile the peak effectiveness was found to increase with increasing grid refinement, which suggests that the underprediction is caused by numerical diffusion errors.

Another feature which is clearly displayed is the overprediction of the strength of the streamwise vortices downstream of the hole. This is consistent with the overprediction of the near wall peak in the V_y - velocity profiles shown in figure 6.8 (b). Unfortunately the variation in the V_y - velocity with increasing grid refinement does not follow consistent trends making it difficult to determine whether the overprediction of

the vortex strength arises from inadequacies in the $k - \epsilon$ turbulence modelling or from numerical error. The examination of the numerical error in the solutions of the y and z - momentum equations by the method of McGuirk and Rodi (1978) has shown that the errors are largely confined to the initial mixing region upstream of $x/D = 2.0$. The results presented in the following section show however, that the overprediction of the vortex strength does not decay with downstream distance but persists unabated to at least $x/D = 40$. This implies that the major cause of the overprediction lies in the turbulence modelling.

A discussion of the inadequacies of the $k - \epsilon$ turbulence model has been presented in section 5.2.4. Perhaps the major shortcoming of the $k - \epsilon$ model in its application to jets in cross-flow is the inability of the isotropic viscosity hypothesis to describe flows with significant curvature. Bradshaw (1973) has observed that curvature effects result in the suppression of the turbulent mixing in the centre of a vortex. In the context of the present work however, the flow curvature associated with the streamwise vortices will result in the enhancement of the turbulent mixing in the vicinity of the wall. Thus the wall shear stress will be underpredicted and with it the rate of decay in the strength of the streamwise vortex.

6.4.3 Surface effectiveness

The overprediction of the streamwise vortex strength has important consequences for the conditions at the wall. This is clearly shown in the predictions of adiabatic wall effectiveness which are compared with the measurements of Eriksen (1971) in figure 6.12. It can be seen that for $J = 2.0$ η_{aw} is generally underpredicted, for $J = 1.0$ the centreline effectiveness is

reasonably predicted but η_{aw} is underpredicted at off-centreline locations and finally for $J = 0.5$ the centreline effectiveness is overpredicted but on off centreline locations η_{aw} is still underpredicted. These discrepancies are not caused by the incorrect prediction of the trajectory of the jet since the velocity profile comparisons presented in section 6.4.1 show that this is well predicted. The major cause of the discrepancies is, as suggested above, the overprediction of the strength of the streamwise vortices. Injection at a velocity ratio, J , of 2 gives rise to large streamwise vortices the overprediction of which results in the entrainment of excess mainstream fluid under the jet causing the underprediction of η_{aw} . The vortices set up by injection at $J = 0.5$ are much smaller and to a large extent are contained within the jet. Consequently little mainstream fluid is entrained into the jet as a result of the vortex action. In this case the vortices act to oppose the spreading of the jet fluid by diffusion. Near the wall the centreline effectiveness is higher than that at off centreline locations so that diffusion acts to transport the conserved scalar away from the centreline. On the other hand the streamwise vortex results in the transport of the conserved scalar towards the centreline. Thus as a result of the overprediction of the streamwise vortices the transport of the conserved scalar away from the centreline is inhibited, causing the overprediction of the centreline effectiveness. This is compounded by the fact that the inadequacies in the $k - \epsilon$ turbulence modelling result in the underprediction of the turbulent mixing near the wall so that the diffusive transport of the conserved scalar away from the centreline will be underpredicted. For $J = 1.0$ the observed discrepancies may be described by a combination of the effects for the $J = 2.0$ and $J = 0.5$ conditions.

6.4.4 Surface heat transfer

Comparisons between the predicted wall heat transfer coefficients and the measurements of Eriksen (1971) for $J = 2.0$, 1.0 and 0.5 are shown in figures 6.13, 6.14 and 6.15 respectively. The measurements were obtained with $T_s = T_\infty$ so that errors in the prediction of the effectiveness field are not reflected in the results. In spite of this only fair agreement has been obtained, with discrepancies of up to 30% between measurement and prediction. For all three conditions the centreline heat transfer coefficient is underpredicted and there is a tendency to overpredict the heat transfer for $z/D > 0.7$. These discrepancies can also be interpreted in terms of the overprediction of the streamwise vortices. At off centreline locations the vortex causes the flow to take on the character of a reattachment region, with which high heat transfer rates are associated; along the centreline the flow separates giving rise to the suppression of the heat transfer rate. The overprediction of the streamwise vortices will result in the exaggeration of these effects. A further cause of the underprediction of the centreline heat transfer rate could be the underprediction of μ_t arising from the combined effects of the variation of C_μ caused by the wake-like character of the flow and the influence of streamline curvature. The latter effect would also cause increases in the heat transfer rate at off centreline locations however it is probable that a major cause of the discrepancies arises from the use of wall functions based on boundary layer flows. Such wall functions assume Couette flow near the wall and are invalid where there are significant velocity components normal to the wall.

6.5 Concluding remarks

Accurate predictions of the jet in crossflow for $0.24 < J < 2.3$ can only be achieved if the development of the flow within the hole is computed. The treatment described in this chapter uses separate calculations of the jet-mainstream interaction and of the flow within the hole which are linked together by assuming appropriate forms of boundary conditions at the hole exit. Two alternative formulations have been employed : a "fixed pressure" form and a "fixed velocity" form. It has been shown that the solutions obtained with the alternative forms are in good agreement and provide accurate representations of the hole exit velocity profile. The rate of convergence of solutions with the fixed velocity form of hole exit boundary condition is superior to that of solutions using the fixed pressure form.

Many features of the flow are well represented by the present predictions. For example, the jet spread and penetration are well predicted and the V_x and V_y velocity profiles generally show good agreement with measured data. However the predictions are subject to numerical diffusion errors which to a large extent are responsible for the underprediction of the peaks in the V_x - velocity and η profiles downstream of the hole. The numerical error is generally confined to the jet and associated shear layers upstream of $x/D = 2.0$ so that the solutions would benefit most from grid refinement in this part of the solution domain.

The comparisons with measured data display discrepancies which are consistent with inadequacies in the $k - \epsilon$ turbulence model and in particular with the isotropic turbulent viscosity hypothesis. This suggests that the use of a turbulence model providing more direct representation of the Reynolds stresses would improve the

accuracy of the predictions. As a result of limitations imposed by computer capacity the adoption of a stress transport model is not justified however, an algebraic stress model should provide the required improvements in the modelling of the influence of streamline curvature and of the wake-like region in the lee of the jet.

The use of a heat flux wall function based on boundary layer type flow with Couette - like flow near the wall is invalid near the jet centreline where there are large velocity components away from the wall. This contributes to the underprediction of the heat transfer coefficients along the jet centreline. The wall functions derived by Chieng and Launder (1980) which have given improvements in the accuracy of wall heat flux predictions for two-dimensional recirculating flows may provide some improvement in the accuracy of the wall heat flux predictions.

CHAPTER 7

LEADING EDGE FILM COOLING CALCULATIONS

7.1 Introduction

In this chapter the three dimensional numerical procedure of Jones (1976) is adapted and applied to the solution of the flow set up by the ejection of fluid from rows of holes in the surface of a cylinder in crossflow. Predictions are obtained for the measured data of Rastogi and Whitelaw (1972) and of the present study, both of which employ this configuration to simulate the flow at the leading edge of film-cooled rotors and stators of gas turbine engines.

The numerical work is restricted to ejection configurations which are symmetric about a plane lying through the stagnation point of the cylinder (see Figure 7.1), since this halves the size of the solution domain required. The initial attempts to model the flow employed cylindrical polar solution grids covering a 180 degree sector from the leading edge to the trailing edge of the cylinder. These were not successful for two reasons :

- i) Core size limitations precluded the use of sufficient grid points to allow the description of both the flow around the cylinder and the details of the interaction between the jet and mainstream flows.
- ii) The cylindrical grid was not suited to the solution of the flow at a distance from the cylinder since the curvature of the co-ordinate system introduces curvature in the dependent variable fields which causes numerical errors (c.f. section 5.2.3).

Two possible solutions to these problems were considered

- i) Co-ordinate transformation
- ii) Embedded grid solutions

The use of co-ordinate transformation based on, say, the streamfunction and velocity potential fields for inviscid flow past a cylinder in crossflow is an attractive approach. In general the flow would be closely aligned with the grid so that, as discussed in section 5.2.3, numerical errors would be minimised. However, this does not overcome the problem that the flow is defined by two length scales; the ejection hole diameter (D) and the radius of the cylinder R . Typically $R/D \approx 7$ making it difficult to obtain an adequate description of both the jet-mainstream interaction (characterised by D) and the mainstream flow around the cylinder (characterised by R), within the maximum number of grid nodes allowed by the available computer resources.

Embedded grid solutions do not suffer from this problem since the solution domains and grid node distributions can be chosen to suit the characteristic length scale of the part of the flow covered by each grid. In this way the grid nodes may be distributed in a much more efficient manner than with a single grid solution reducing the total core storage requirement of the solution. By suitable choice of grids it is also possible, with the embedded grid solutions, to arrange so that much of the flow is aligned with the calculation mesh, minimising the numerical errors of the solution. In view of these arguments embedded grid solutions have been employed for the work presented in this chapter.

The following section of the chapter is a description of the computational details of the embedded grid solution, including the validation of this technique for a cylinder in crossflow

without ejection. This is followed by the presentation and discussion of the predictions of the measured data of Rastogi and Whitelaw (1972) and of the present study for cylinders with ejection. Finally, concluding remarks are given in section 7.4.

7.2 Computational details

The arrangement of the solution meshes has been chosen so that the flow is aligned as closely as possible with the mesh, thus reducing numerical errors. At a large distance from the cylinder the flow is only slightly perturbed by the presence of the cylinder and the approximately uniform freestream is best represented by a rectangular grid. On the other hand in the vicinity of the cylinder it is important to resolve the details of the near-wall flow. This is best achieved by a cylindrical polar grid. A suitable grid arrangement is therefore a cylindrical grid embedded within a rectangular grid as shown in figure 7.2. To further reduce the cylindrical grid solution domain required, use has been made of a version of Jones (1976) method which has been extended by Priddin (1979) to allow wall boundaries which do not conform to the grid. It is therefore possible to use the rectangular grid solution to represent the flow downstream of the region of interest.

The cylindrical grid and rectangular grid are arranged to overlap to allow transfer of information between the grids and to ensure continuity of gradients of quantities across the interface.

7.2.1 Treatment of the rectangular grid cells at the interface

The treatment of rectangular grid cells which interface with the cylindrical grid solution has two basic features :

- 1) The calculation of the area mean flux of quantities across the cell faces at the interface by linear interpolation from the cylindrical grid solution and the modification of the rectangular grid interface cell coefficients in accordance with the calculated flux.
- 2) The specification of fixed values at rectangular grid nodes within the region enclosed by the interface to ensure the correct computation of source terms for the cells at the interface.

Proper account must be taken of the staggered grid arrangement (see figure 4.1) in order to define the interface cells for V_x and V_y . Accordingly we start by defining the V_x and V_y cell faces which lie just within the region bounded by the $N_r = 1$ and $N_\theta = 1$ grid lines of the cylindrical grid. This defines a preliminary interface for the V_x and V_y cells which ensures an overlap of the grids. Within this region nodes are chosen at which the V_x and V_y velocities are to be fixed. These arrangements are shown in figure 7.3 (a) and (b) for the V_x and V_y grid cells respectively. The interface for the V_z velocity and scalar grids is defined by the cell faces on which the V_x and V_y velocities are fixed. This boundary is shown in figure 7.3 (c). It can be seen that some of the fixed value nodes for V_x and V_y lie outside the interface. The final V_x and V_y grid interface is obtained by modifying the preliminary arrangement to remove these points. This ensures consistency between the interface definitions of the staggered grids and the scalar grid. Finally, the fixed value node locations are updated to ensure the correct computation of source terms.

Once the interface and fixed value node locations have been defined, the fluxes and values are determined as follows :-

- i) Transform the cylindrical grid V_r and V_θ velocity components to V_x and V_y components of the rectangular grid.
- ii) Calculate the area mean mass flux $(\hat{m}_I)$ and the convective flux of ϕ , $(\hat{m}\phi)_I$, across each cell face on the interface by linear interpolation from the cylindrical grid solution.
- iii) Determine ϕ at the fixed value node locations using linear interpolation from the cylindrical grid solution. (Note that where ϕ is V_x or V_y an area meaned value on the plane perpendicular to the velocity component is employed).

The following boundary conditions are employed for the rectangular grid solution cells bounding the interface :-

For V_x and scalars the coefficients of the cells bounding the interface are modified by setting $A_I^\phi = 0$ and modifying the source terms as follows :-

$$b^\phi = b^\phi + \Delta b^\phi \quad (7-1)$$

$$c^\phi = c^\phi + \Delta c^\phi \quad (7-2)$$

Where the flow leaves the cell through the interface face $\Delta b^\phi = \Delta c^\phi = 0$, and

$$\Delta b^\phi = - |\hat{m}_I| \quad (7-3)$$

$$\Delta c^\phi = \frac{\hat{m}_I}{|\hat{m}_I|} (\hat{m}\phi)_I \quad (7-4)$$

where the flow enters the cell through the interface face.

In the present work the grid Peclet numbers in the x and y directions of the rectangular mesh at the interface are always

greater than 2. Consequently upwind differencing is employed for these calculation cells and diffusive terms do not enter the finite difference equations (see the hybrid difference expressions equations (4-27) and (4-28)). Thus the modelling of the convective terms by equations (7-1) to (7-4) provides a sufficient description of the fluxes through the rectangular grid cell faces on the interface.

Since the velocities are fixed on the interface, the correct boundary condition on the pressure correction is $A_I^p = 0$ through equation (4-36), together with the use of the mass flux $\hat{m}_x$ in the calculation of $\dot{m}_p^*$.

Application of the procedure for V_z and scalars to V_x and V_y results in inconsistent solutions to the pressure correction equation. This arises because the assumption of a fixed flux across the interface of the staggered grid cells of the velocity components normal to the interface is only correct if $p' = 0$ for the scalar grid cell bounding the interface. In general this will not be satisfied and as a result the boundary condition on the momentum equations for velocity components normal to the interface makes use of the fixed velocity points.

7.2.2 Determination of boundary values at the interface for the cylindrical grid

The planes defined by $r(N_r)$ and $\theta(N_\theta)$ form the boundaries of the cylindrical grid solution which lies within the rectangular grid solution. The boundary condition on the $r(N_r)$ plane is formed by interpolating values from the rectangular grid solution using linear interpolation. The $\theta(N_\theta)$ boundary forms an exit boundary on the cylindrical grid solution and its treatment is similar to that employed for the fixed velocity form of the hole

exit boundary condition described in chapter 6, i.e. V_θ is specified by interpolation from the rectangular grid solution, whereas boundary values of V_r , V_z , k , ϵ and T are obtained by linear extrapolation from within the cylindrical grid solution. A consequence of the coarseness of the rectangular grid is that the distribution of V_θ close to the wall is not well represented. The numerical diffusion which results leads to the overprediction of the thickness of the boundary layer on the cylinder. The situation could be improved by further grid refinement in this region however computer storage limitations did not allow this. In order to overcome this problem the V_θ boundary condition is formed by interpolation from the rectangular grid up to the edge of the apparent boundary layer and blending between this point and the wall a profile with the same shape as this part of the V_θ profile at $\theta (N_\theta - 1)$.

The system of embedding the cylindrical grid solution within that of the rectangular grid, described in this section and the previous one, introduces errors into the solutions. The method of specifying the fluxes through the faces of the rectangular grid cells at the interface mimics the operation of the upwind difference scheme and, apart from numerical diffusion errors associated with the upwind difference approximation, does not degrade the accuracy of the solution. Interpolation errors also arise in the determination from the rectangular grid solution of the boundary values on the $r (N_r)$ and $\theta (N_\theta)$ planes of the cylindrical grid. Since the interpolation has the same order of accuracy as the finite difference approximation then the interpolation errors will be of similar magnitude to the numerical diffusion error of the rectangular grid solution. The nearer the interface is located to the surface of the cylinder the

greater is the curvature of the flow which has to be resolved by the rectangular grid solution and the numerical diffusion errors increase. Consequently it is necessary to locate the interface sufficiently far from the surface of the cylinder for the numerical error of the rectangular grid solution to be small. This has been achieved by computational experiment.

The approximation involved in the treatment of the near wall region of the exit boundary of the cylindrical grid does introduce errors, however, the calculations completed to validate the solutions have shown that the error is localised and small (c.f. section 7.2.4).

7.2.3 Validation of the rectangular grid solution

The initial validation of the rectangular grid solution was completed by predicting the laminar flow about a cylinder in crossflow at a Reynolds number based on the cylinder diameter of 40. The domain employed for these solutions is shown in figure 7.4. A symmetry boundary condition is assumed at the $Y(1)$ boundary and the velocities at the $X(1)$ and $Y(N_y)$ boundaries are specified by the analytic solution for a cylinder in crossflow. At the $X(N_x)$ boundary the condition $\partial\phi/\partial x = 0$ is applied as with previous exit boundary conditions. The location of the boundaries as determined by computational experiment are :-

$$X_u / R = 3$$

$$X_D / R = 8$$

$$H / R = 7$$

Solutions obtained on grids of 19×13 and 25×20 nodes ($N_x \times N_y$) gave results which were virtually identical. The predicted length of the recirculation downstream of 2.13 cylinder

diameters is in excellent agreement with the observations of Taneda (1956).

A number of modifications were made to prepare the rectangular grid solutions for the predictions presented in section 7.3.

- i) The flow around the cylinder becomes turbulent at the ejection point. It is appropriate to include the equations for k and ϵ in the solutions.
- ii) In order to account for the presence of the windtunnel wall the wall boundary condition (equations (4-44), (4-47) and (4-48)) is specified at $Y(N_y)$ and the $X(1)$ boundary condition modified to account for the boundary layer on the wall.
- iii) To reduce the computational expense an afterbody was fitted to the rear of the cylinder as shown in figure 7.5. This allows a reduction in the downstream extent of the solution domain (to $X_D/R = 2$) and also significantly enhances the convergence of the solution by suppressing the recirculation downstream of the cylinder. Comparisons of predictions obtained with the afterbody with those without the afterbody show very little effect on the flow up to $\theta \approx 75$ degrees.

A refined version of the 25×20 node grid employed in the laminar flow solution, with $20 \times 25 \times 8$ nodes ($N_x \times N_y \times N_z$) is employed for the work presented in section 7.3. This grid was the finest that could be employed on the available computer facilities.

7.2.4 Validation of the embedded grid solutions for a cylinder without ejection

The initial evaluation of the embedded grid solution was carried out for a cylinder in crossflow at a Reynolds number based on diameter of 40 and also to compare with the measurements of the present study at a Reynolds number of 8.5×10^4 . The sequence of operations employed for the embedded grid calculations is summarised in table 7.1.

Step	Operation
1	Obtain a solution for the rectangular grid alone and file the result.
2	Interpolate to obtain boundary values for the cylindrical grid (c.f. section 7.2.2).
3	Retrieve any previously filed cylindrical grid solution (or initialise) and compute for several iterations (~ 50) using the new boundary values. File the result.
4	Calculate the fluxes etc. for the interface cells of the rectangular grid (c.f. section 7.2.1).
5	Retrieve the filed rectangular grid results and continue the solution using the new fluxes etc. File the result.
6	Test for convergence and repeat the sequence from 2 as necessary.

Table 7.1 Sequence of operations for the embedded grid computations.

The convergence criteria used were the satisfaction of the residual mass flux condition, equation (4-39) for both the rectangular and cylindrical grid solutions and that the changes in the velocity and η fields of the cylindrical grid solution on successive completions of step 3 of the sequence be less than 1% V_∞ and 0.005 respectively.

The $r - \theta$ domain employed for the cylindrical grid solutions is shown in figure 7.6. The extent of the domain employed :

$$r(N_r) / R = 2.0$$

$$\theta(N_\theta) = 76$$

is based on the domain study carried out in section 7.2.5, as was the finite difference mesh :

$$\text{Computation for } Re = 40 : \quad N_r \times N_\theta = 20 \times 20$$

$$\text{Computation for } Re = 8.5 \times 10^4 : \quad N_r \times N_\theta = 21 \times 25$$

Both solutions were set up for laminar flow. This is appropriate since, through a combination of the acceleration around the surface and curvature effects, the boundary layer remains laminar over the major part of the upstream surface of the cylinder.

The embedded grid solution for $Re = 40$ is compared with a Blasius series solution matched to the potential flow solution for a cylinder in crossflow in figure 7.7. The solutions agree to within 10%. A comparison with the velocity profile measurements of the present study is shown in figure 7.8. Again, agreement between the prediction and measurements is generally within the experimental error, the largest discrepancy (10%) occurring in the V_θ profile at $\theta = 76$ degrees. This discrepancy is caused by the use of an afterbody in the prediction and the assumptions

made in describing the V_θ profile on the exit plane of the cylindrical grid solution (c.f. section 7.2.2).

Accurate predictions of the pressure field are difficult to obtain since numerical errors in the solutions of all the momentum equations are reflected as errors in the pressure. A comparison between the predicted and measured static pressure distribution on the surface of the cylinder is shown in figure 7.9. In general good agreement has been obtained, however, there are discrepancies in C_p of up to 0.10 for $\theta < 10$ degrees and 0.75 for $\theta \sim 80$ degrees. The latter discrepancy is consistent with the overprediction (by 10%) of V_θ near the wall at $\theta = 76$ degrees and reflects the influence of the afterbody and the exit boundary assumptions as described above.

The error in C_p for $\theta < 10$ degrees is equivalent to an error in the prediction of the velocity magnitude of about 5% V_∞ . Part of this error arises as a result of incomplete convergence of the iteration sequence between the two solutions. Six cycles of the solution sequence given in table 7.1 were employed for the calculation and inspection of the solution suggest that approximately 2% of the error arises from incomplete convergence. An examination of the numerical diffusion errors of both the cylindrical and rectangular grid solutions using the method of McGuirk and Rodi (1978) has shown that, although the errors are small, in neither solution is the error negligible. Thus the remaining error (3% V_∞) arises as a result of numerical diffusion in both the rectangular and cylindrical grid computations and errors in the interpolations between the grids.

7.2.5 Validation of the cylindrical grid solution with ejection

The boundary conditions employed in the cylindrical grid solutions are summarised in table 7.2 and the solution domain is shown in figure 7.10. The solutions include the calculation of the flow within the ejection hole using an arrangement similar to that employed in chapter 6 but with the appropriate transformation of the velocities at the hole exit to the cylindrical grid co-ordinates.

Boundary	Condition
$r(1)$	$\left\{ \begin{array}{l} \text{Hole exit : Fixed velocity form} \\ \text{(c.f. section 6.3.2.)} \\ \text{Wall : Eqns. (4-44), (4-47) and (4-48),} \\ \delta\eta / \delta r = 0 \end{array} \right.$
$r(N_r)$	Specified from rectangular grid solution
$\theta(1)$	Symmetry : $V_\theta = 0$; $\delta / \delta \theta = 0$
$\theta(N_\theta)$	Specified from rectangular grid solution
$z(1); z(N_z)$	Symmetry : $V_z = 0$; $\delta / \delta z = 0$

Table 7.2 Boundary conditions for the cylindrical grid solution.

Boundary locations for Θ (N_Θ) of between 75 and 80 degrees are employed, the maximum consistent with the region largely unperturbed by the presence of the afterbody. Other boundary locations are fixed by the geometry of the experiment with the exception of r (N_r) which must be located sufficiently far from the surface of the cylinder to ensure that the errors involved in the interpolations between the grids and the solution on the rectangular grid are acceptably small. The location of the r (N_r) boundary was determined for several ejection configurations by computational experiment and in each case the same result, r (N_r)/ $R = 2$, was obtained.

The results of grid dependence tests on the cylindrical grid solution with ejection are shown in figure 7.11. The $20 \times 24 \times 18$ node ($N_r \times N_\Theta \times N_z$) grid was the finest that could be used on the available computer facilities and it can be seen that grid independent solutions have not been obtained. A marked sensitivity of both the velocity and η profile to grid refinement is apparent.

The numerical accuracy of the cylindrical grid solution has been studied further using the method of McGuirk and Rodi (1978), described in chapter 5. This has shown that numerical diffusion is greatest in the vicinity of the secondary jet and its associated shear layers and that outside this region numerical diffusion is generally small. The results for the r - momentum, Θ - momentum and conserved scalar equations of the $\Theta_r = 40$ degree configuration of the present study are shown in figure 7.12 (a), (b) and (c) respectively. The large positive peaks in the residuals for the r and Θ - momentum equations imply the underprediction of the r - momentum of the secondary jet and of the Θ momentum

for $\tilde{r}/R < 1.13$. Once again, numerical diffusion is most significant for the conserved scalar and will result in the overprediction of the diffusion of the secondary jet fluid into the mainstream.

The predictions presented in section 7.3 make use of grids based on the $20 \times 24 \times 18$ node ($N_r \times N_\theta \times N_z$) mesh of the grid dependence study. The solutions require between 7 and 10 cycles of the calculation sequence given in table 7.1 and a total CPU equivalent to between 3 and 4 hours on an IBM 370-168, 75% of which is expended on the cylindrical grid solution. The maximum core storage required is 900k bytes.

7.3 Comparison with experimental measurements

The configurations chosen for prediction have been restricted to ejection geometries which allow the use of symmetry boundary conditions at $\theta(1)$, $z(1)$, and $z(N_z)$. This minimises the cylindrical grid solution domain size and the computational effort required. The experimental work of the present study and of Rastogi and Whitelaw fall into this category.

7.3.1 Measurements of Rastogi and Whitelaw (1972); $\theta_j = 0^\circ$

In this study measurements of η_{iw} on the jet centreline were obtained at two density ratios ($\rho_j/\rho_\infty = 1.0$ and 2.1) and velocity ratios up to $V_j/V_\infty = 2.4$. As discussed previously in chapter 3 bistable flow was observed during the experimental work of the present study, for $\theta_j = 0$ and $J > 1.0$ when the splitter plate was not fitted. Rastogi and Whitelaw comment on the sensitivity of the flow for low θ_j , but do not seem to have observed the bistable flow.

Figure 7.13 contains comparisons between the measured and predicted η_{iw} for conditions at which bistable flow is not expected (i.e. Momentum Ratios $\rho_{\tau} V_{\tau}^2 / \rho_{\infty} V_{\infty}^2 < 1.0$). In general, the agreement is better than the quoted experimental error ($\pm 3\%$ of unity). The comparisons with $\rho_{\tau} / \rho_{\infty} = 2.1$ show slightly worse agreement than those at $\rho_{\tau} / \rho_{\infty} = 1.0$. This possibly arises from the influence of buoyancy on the turbulent Prandtl/Schmidt number (see Gibson (1978)) which is assumed to be constant in the predictions.

The predictions of this data were completed prior to the experimental work presented in chapter 3 and include conditions with $\rho_{\tau} V_{\tau}^2 / \rho_{\infty} V_{\infty}^2$ for which bistable flow would be expected. Predictions at these conditions produce a totally different flow regime to that found at lower momentum ratio conditions as shown in figure 7.14. In the calculations the position of the contra-rotating vortex pair set up across the front of the cylinder for $\rho_{\tau} V_{\tau}^2 / \rho_{\infty} V_{\infty}^2 > 1$ is fixed by the imposition of the symmetry plane boundary condition at $\Theta = 0$. No such constraint exists in the real flow and the vortices are free to build and shed in an unsymmetrical manner. We may attempt to understand the behaviour observed in the experimental work of the present study in terms of the interaction of the vortices shed from the ejection point with those at the rear of the cylinder. With the splitter plate fitted the shed vortex interacts with the stationary vortex downstream of the cylinder increasing its overall vorticity. Since the splitter plate fixes the flow direction this leads to an increase in the flow on this side of the cylinder, moving the upstream stagnation point to the opposite side of the jet and shedding a vortex on the opposite side of the cylinder. Hence the unstable behaviour observed with the splitter plate fitted.

Without the splitter plate, the vortex shed from the front of the cylinder interacts with the vortex street behind the cylinder. Because the flow direction is no longer constrained however, this does not influence the flow on the upstream surface of the cylinder, leading to the bi-stable flow behaviour observed without the splitter plate fitted.

It is worth mentioning here that Warren (1960) has observed a stable flow akin to that shown in figure 7.14 for $\rho_s V_s^2 / \rho_o V_o^2 > 1$ in connection with the cooling of the nose of axisymmetric bodies in hypersonic flight by the ejection of fluid at the stagnation point. In this case however a vortex ring is formed around the ejection point, and the instabilities observed above arising from vortex shedding do not occur.

7.3.2 Measurements of the present study

Comparisons between the measured and predicted velocity profiles for $\theta_s = 0^\circ$, 20° , and 40° are shown in figures 7.15, 7.16 and 7.17 respectively. Over the major part of most of the velocity profiles the predictions agree with the measurements to within the experimental error, however, there are regions where discrepancies arise. For example, there is a tendency to underpredict V_θ for $r/R < 1.3$ which is particularly marked in the region of the velocity peak immediately downstream of the ejection point for $\theta_s = 20^\circ$ and 40° . This is a consequence of the numerical diffusion in the solutions of the r and θ momentum equations discussed in section 7.2.5.

The $k-\epsilon$ turbulence modelling employed here makes use of the assumption that the flow is fully turbulent. In practice, although the flow is certainly turbulent downstream of the

ejection point, it is laminar upstream of this point. The turbulent boundary layer predicted upstream of the ejection point is thicker than the corresponding laminar boundary layer of the actual flow. The additional boundary layer turbulence is unlikely to influence the flow downstream of the ejection point significantly, however, the increased boundary layer thickness does have a small effect. This is most noticeable for $\Theta_j = 40$ degrees where it has caused the underprediction of the strength of the streamwise vortices downstream of the holes.

Concentration profile comparisons for $\Theta_j = 0, 20$ and 40 degrees are shown in figures 7.18, 7.19 and 7.20. It can be seen that, for $\Theta_j = 0^\circ$, the prediction indicates that the jet reattaches to the surface of the cylinder well upstream of the reattachment point shown in the measurements. Consequently the calculated surface concentration (η_w) is in poor agreement with measurement.

The predictions of the data of Rastogi and Whitelaw (1972) for a similar configuration are in good agreement with the measurements however (see figure 7.13). These apparently inconsistent results appear to arise as a result of differences in the numerical diffusion errors of the solutions. Comparison of the truncation errors of the solutions for the $J = 0.87$ condition of Rastogi and Whitelaw and the $J = 0.89$ condition of the present study has shown that the numerical diffusion error in the outer region of the jet-mainstream interaction is greater in the latter solution. The diffusion of jet momentum into the mainstream flow which results is the probable cause of the early reattachment of the jet to the cylinder. Now the cylindrical grid solution mesh has been chosen to scale on the ejection hole diameter, D . In terms of the radius of the cylinder, R , however, the grids employed for the predictions

of the present work are coarser than those employed the data of Rastogi and Whitelaw since in the former $R/D = 12.5$ and in the latter $R/D = 6.25$. The larger numerical errors associated with the prediction of the present results reflect the strong influence of the cylinder on the flow for $\Theta_j = 0^\circ$ as discussed in section 3.3.3. A notable feature of figures 7.19 and 7.20 is the underprediction of the radial spread of the jet fluid. Consideration of the velocity profiles in this region shows that $\partial(\bar{v}_\theta)/\partial r$ is negative and the discrepancy is clearly a result of the influence of streamline curvature on the turbulent mixing. As a consequence of a combination of the underprediction of the radial spread of the jet and numerical error, the lateral spread of the jet tends to be overpredicted. This is shown clearly in the wall concentration results for $\Theta_j = 20^\circ$.

Finally, having considered errors arising from the turbulence modelling and numerical aspects of the solutions, the measurements are not entirely blameless. For example, the V_r - velocity profiles immediately downstream of the hole for $\Theta_j = 20^\circ$ and 40° exhibit a small negative peak in the region of $r/R = 1.15$. This is an artefact of the steepness of the gradient $\partial v_\theta/\partial r$ and the inadequate spatial resolution of the 5-hole probe.

7.4 Concluding remarks

The embedded grid system described in this chapter has allowed the computation of the ejection of fluid from the surface of a cylinder in cross-flow with much greater accuracy than would have been possible using a single cylindrical polar grid. The essential features of the embedding procedure are not specific to the present problem and may find application to other problems in fluid dynamics where a relatively small region of particular interest

lies within a flow field of much greater extent.

The calculated results are in encouraging agreement with measurement, however, in spite of the use of embedded grids further grid refinement is necessary to reduce numerical errors. Alternatively the application of numerical methods based on higher order difference approximations (e.g. Raithby (1976b) and Leonard (1979)) would have potential for improving the numerical accuracy of the solutions.

The comparisons also show the need for changes in the $k - \epsilon$ turbulence modelling to account for the laminar flow on the surface upstream of the ejection point and the influence of curvature on the turbulent mixing. In principle the low Reynolds number form of the $k - \epsilon$ model, Jones and Launder (1973), is capable of treating the combination of laminar and turbulent flows on the cylinder. As discussed in chapter 5, however, this requires a very much finer solution mesh than that employed here and is not yet a practical proposition. A simpler approach that may prove adequate would be to retain the high Reynolds number form of $k - \epsilon$ model but to neglect the turbulent contribution to momentum transfer upstream of the ejection point.

The accuracy of the prediction of $\eta_{1,2}$ is of major concern to the blade cooling engineer. In the comparisons with the experimental data of the present study it is clear that the accuracy of the predictions of $\eta_{1,2}$ would be improved by the inclusion of streamline curvature effects in the modelling of turbulence. The applicability of turbulence modelling based on an isotropic turbulent viscosity defined by equation (4-16) to complex three-dimensional flows with streamline curvature has been discussed in chapter 6. Again, the adoption of a full Reynolds

stress closure is not justified since the numerical error of the solutions is significant, however, a relatively ad-hoc treatment of curvature effects, possibly based on the work of Priddin (1975), deserves attention and should prove adequate for engineering calculations.

CHAPTER 8

CLOSURE

8.1 Introduction

In this chapter the major conclusions of the present work are reviewed and recommendations made for the direction of future work, with an emphasis on understanding the behaviour of film cooling on gas turbine blades. The chapter is divided into two sections which consider experimental aspects of the work and theoretical modelling respectively.

8.2 The experimental study of film cooling

8.2.1 Major conclusions

The experimental work of the present study concerned the determination of the behaviour of film cooling at the leading edge of a gas turbine blade. Although the heat transfer aspects of the flow have been studied previously (c.f. section 1.3.3) the present work includes the first quantitative measurements of the aerodynamics of leading edge cooling configurations.

The results presented in chapter 3 for ejection from rows of holes on a cylinder in crossflow with $\Theta_x = 0, \pm 20$ and ± 40 degrees show that the flow behaves quite differently for ejection at ± 20 and ± 40 degrees than for ejection at $\Theta_x = 0$ degrees. In the ± 20 and ± 40 degree configurations the mainstream flow has a significant crossflow component across the exit of the hole to which the film cooling jet presents a blockage. This is similar to the situation arising in other jet in crossflow configurations so that, although the details of the flowfield are affected by streamline curvature and pressure gradients, the flow characteristics are essentially similar. This is not the case for ejection

at $\Theta_j = 0^\circ$ however, since (for $J < 1$ at least) the mainstream has a negligible crossflow component at the hole exit and, in comparison to the cylinder, the film cooling jet presents little blockage to the mainstream flow. This is reflected in the measured velocity profiles which show little distortion arising from the injected flow. In contrast the velocity profiles for the ± 20 and ± 40 degree ejection configurations are significantly distorted.

Bistable flow was observed for ejection along the stagnation line with $J \gg 1.0$. A possible mechanism for this behaviour is proposed in section 7.3.1 which is based on the shedding of spanwise vortices set up by the ejection, and their interaction with the spanwise vortices at the rear of the cylinder.

The ± 20 and ± 40 degree ejection configurations may be characterised approximately in terms of the velocity ratio, J_L , based on the mainstream velocity in the locality of the ejection point. Comparisons with measurements of film cooling on flat plates with zero pressure gradient flow show that, at a given J_L , the present results have decreased jet penetration and this is attributed to a combination of the effects of the radial component of the mainstream flow and the acceleration.

One of the major problems associated with the cooling of gas turbine blade leading edges concerns the high heat transfer coefficients which occur along the stagnation line. Ejection of coolant on the stagnation line would appear to be beneficial since the stagnation line may, in part, be lifted away from the surface, and at the same time the ejected air forms a protective film on the surface. The results of the present work suggest several problems with this approach however :

- 1) It is difficult to obtain a symmetrical division

of the coolant to either side of the stagnation line for $\rho_s V_s^2 / \rho_\infty V_\infty^2 < 1$, and impossible for $\rho_s V_s^2 / \rho_\infty V_\infty^2 \geq 1$.

- 2) Although the aerodynamic losses associated with the stagnation line ejection are small, so are the levels of effectiveness.
- 3) As a result of the lift on the blade, the stagnation streamline of the mainstream flow is curved. This will cause the ejected flow to be biased towards the suction surface of the blade.
- 4) The rotor blade experiences severe negative incidence as it passes through the wakes of the upstream stator blades, which will tend to push the ejected flow onto the pressure surface.

In view of (3) and (4) it is debatable whether a stagnation line ejection arrangement could significantly reduce the time-mean heat transfer coefficient at the leading edge by lifting the stagnation line off the surface. This, combined with the low surface effectiveness levels, makes the configuration unattractive for blade cooling purposes.

The $\Theta_s = \pm 20$ and ± 40 degree configurations are more promising since there is no doubt over the direction of the coolant flow and the surface effectiveness levels offer significant potential for cooling the surface downstream of the holes. There are also indications from engine measurements that the use of cooling air from discrete holes located away from the stagnation line modifies the flow in the stagnation region to reduce the convective heat transfer. Further cooling of the stagnation region is provided by conductive heat transfer. The present measurements

show, however, that there is a price to be paid for the improved cooling given by holes located away from the stagnation line and that is a significant increase in the aerodynamic losses arising from the mixing between the coolant and mainstream flows. The present measurements suggest that, for a given cooling duty, there will be an optimum location for the ejection point which reduces the combined aerodynamic and thermodynamic cycle loss arising from the coolant flow to a minimum.

8.2.2 Future work on leading edge film cooling

Additional measurements on the configurations employed in the present study should include aerodynamic and heat (mass) transfer measurements at a density ratio (ρ_s/ρ_∞) of 2, representative of the conditions in gas turbine engines. The measurement of the heat transfer coefficient distribution with ejection is also required. In several studies heat transfer coefficient measurements with film cooling have been obtained using test plates with electrical heater strips mounted on an electrically and thermally insulating substrate, see for example Eriksen (1971) and Foster (1973). Measurements obtained by this method allow the major effects of film cooling on surface heat transfer to be determined, however, the test conditions are not representative of gas turbine engines. Alternatively, short duration heat transfer measurement techniques (e.g. Smith et al (1974)) would allow measurements to be made at wall to freestream and coolant to freestream temperature ratios representative of engine conditions.

The present work has employed film cooling holes normal to the surface. These have two undesirable characteristics : a large aerodynamic loss which increases as θ_j increases and a

small lateral spread of coolant on the surface. Hartsel (1970) found that the aerodynamic loss associated with film cooling was decreased by inclining the holes in the streamwise direction. This is not desirable at a leading edge since the conductive cooling of the stagnation region would be reduced. Inclining the holes in the spanwise direction reduces the aerodynamic loss, Hartsel (1970) and increases the lateral spread of coolant on the surface, Taylor and Whitelaw (1975). As a result of these benefits spanwise inclined holes have been extensively employed for leading edge film cooling. The leading edges of the first stage stator blades of the high pressure turbine are cooled by multiple row arrays of film cooling holes. Whilst the heat transfer performance of spanwise inclined and multiple row leading edge film cooling configurations have been investigated, Hanus and L'Ecuyer (1976) and Sasaki et al (1976), the aerodynamics of these configurations has not been investigated.

In order to simplify the theoretical modelling the present study has concentrated on ejection geometries which are symmetric about the stagnation line. In practice, neither the ejection geometry nor the mainstream flow are symmetric about the stagnation line and the influence of these asymmetries needs to be investigated.

8.3 The numerical modelling of film cooling

8.3.1 The present contribution

Procedures have been developed for the calculation of film cooling from angled flush slots, a circular jet flowing perpendicularly to a crossflowing stream and film cooling at a leading edge of a gas turbine blade. The work on two dimensional flush

slots has shown that for $J > 0.6$ the flowfield set up by the ejection is dependent on the velocity profile at the slot exit. As a result of the pressure field set up by the interaction between the jet and mainstream flows the velocity profile of the ejected flow becomes distorted. A simple treatment based on the assumptions of constant total pressure in the ejected flow and constant angle of ejection has been derived to form a boundary condition at the exit of the slot. Provided errors from other sources are insignificant the treatment has allowed the prediction of the velocity and temperature fields to within the experimental error of the available measurements. For $J > 1$ regions of significant numerical diffusion arise in the shear layers bounding the jet and, for $\alpha = 30$ and 60 degrees, at the slot exit. The numerical diffusion leads to the underprediction of the reattachment length downstream of the slot (by 20% at $\rho_{\infty} V_{\infty}^2 / \rho_{\infty} V_{\infty}^2 = 1.40$) and the overprediction of η_{aw} by up to 30%.

A more detailed examination of the influence of the distortion of the flow within the hole has been completed for the computations of the jet in crossflow. It has been found that the flowfield is sensitive to the form of the hole exit velocity profile for $0.5 \leq J \leq 2.0$, the maximum sensitivity occurring for $J = 1.0$, and that it is necessary to compute the development of the flow within the hole to obtain accurate solutions of the flowfield. The method that has been developed employs separate, linked, computations of the flow within the hole and of the jet-mainstream interaction and provides solutions of the velocity field at the hole exit to within 7% V_{∞} . The predictions of the V_x and V_y velocity profiles and of the jet spread and penetration show generally good agreement with measured data however numerical

diffusion errors in the initial mixing region of the jet result in the underprediction of the peaks in the V_x velocity and η profiles downstream of the hole (by 17% and 0.10 respectively for $J = 2.0$).

The solution of the flowfield arising from the ejection at the leading edge of a gas turbine blade has necessitated the resolution of a flow characterised by two length scales : the film cooling hole diameter, and the leading edge radius. This problem has been solved by the development of the embedded grid system described in chapter 7 in which a fine mesh computation of the jet-mainstream interaction is embedded within a coarser mesh solution for the mainstream flow around the leading edge. The embedded grid solutions have been validated by the computation of the flow around a cylinder in crossflow at Reynolds numbers (based on diameter) of 40 and 8.5×10^4 . This has shown that the velocity field may be predicted typically to within 4% V_∞ , the errors arising from a combination of numerical diffusion in the rectangular and cylindrical grid solutions and errors in the interpolation between the grids. A localised region of up to 10% error in the velocity field occurs at the exit boundary of the cylindrical grid as a result of approximations in the formulation of the exit boundary condition.

The computed results with ejection are in encouraging agreement with measurement, however, further refinement of the cylindrical grid is necessary to reduce numerical diffusion which results in the underprediction of the peaks in the V_θ - velocity and η - profiles downstream of the ejection point.

The high Reynolds number form of the k - ϵ turbulence model has been employed for all the turbulent flow computations completed

for the present work. In the computations of film cooling from a two-dimensional flush slot the turbulence model performed satisfactorily. One shortcoming has been identified however, the underprediction of μ_t in the wake-like region far downstream of the slot which resulted in too slow a decrease in η_{aw} with downstream distance for $X/S > 20$ and $J > 1$. Inadequacies in the turbulence model are more apparent in the solutions for the jet in crossflow. The predictions of η_{aw} were significantly in error due to the overprediction of the strength of the streamwise vortices as a result of the use of the isotropic turbulent viscosity hypothesis in the turbulence modelling. In the case of the leading edge film cooling, the flowfield includes regions of significant streamline curvature together with laminar boundary layers upstream of the ejection point and turbulent flow downstream. The inability of the turbulence model to cope with these aspects of the flow results in the underprediction of the penetration of the ejected flow, shown in the concentration profiles, and errors in the surface concentration results.

Calculations of wall heat flux are in only fair agreement with measurement and exhibit errors of up to 40%. Although the discrepancies may be partly attributable to the turbulence modelling the wall functions employed also appear to be at fault.

Current methods of film cooling system design make use of laterally averaged information and no attempt is made to account for the three-dimensional nature of film cooling from rows of discrete holes. In terms of laterally averaged quantities the numerical methods developed in the course of the present work provide predictions of aerodynamics and wall heat transfer coefficients to an acceptable level of accuracy for blade cooling

design. The usefulness of the methods would, however, be greatly enhanced by improvement in the accuracy of the predictions of η_{aw} .

8.3.2 Recommendations for future work

Future work on the methods developed in the course of the present study should take place in two directions :

- 1) Improvement of the methods in order to increase the accuracy of the calculated results.
- 2) Use of the methods for blade cooling design.

Improvement in the definition of the peaks in the velocity and effectiveness profiles downstream of the ejection point would be obtained by an increase in the numerical accuracy of the solutions. In the short term the technique described by McGuirk, Taylor and Whitelaw (1981) which provides guidance for grid refinement should give improvements in numerical accuracy. In the longer term the present active work on improved difference schemes (e.g. Raithby (1976) and Leonard (1979)) will hopefully result in a scheme which is both more accurate and as reliable as hybrid differencing. At the same time increases in computer capacity should allow further refinement of the grids which can be employed.

The isotropic turbulent viscosity hypothesis employed in the $k - \epsilon$ turbulence model does not appear to provide an adequate description of the turbulent stresses arising from ejection from a single hole and rows of holes. The adoption of a stress-transport closure is not justified since the additional computer resources required would necessitate the use of coarser computational meshes, increasing the numerical diffusion error of the solution. Algebraic stress models have been shown to provide a

more accurate description of turbulent stresses for curved wall boundary layers and separated flows than the $k - \epsilon$ model, Rodi (1981). The additional computer resources required by the adoption of an algebraic stress turbulence model are relatively small so that this may provide the required improvement in the turbulence modelling without degrading the numerical accuracy of the solution.

The accuracy of the wall heat flux predictions may be improved by the adoption of modified wall function described by Chieng and Launder (1981) which has given improved predictions of wall heat flux for a two-dimensional recirculating flow. It is not clear whether this wall function would improve the predictions of wall heat flux for the three-dimensional computations however.

The present work has not included the prediction of the wall heat flux for leading edge film cooling configurations. These results would be of interest, however, and the predictions of wall heat flux would allow the influence of ejection from rows of holes located away from the stagnation line on the convective heat transfer to the stagnation region to be determined.

The application of the numerical procedures to blade cooling design would best be served by the use of the methods in a parametric study to build up a data base for design. In use the numerical method would thus become analogous to a windtunnel and the predicted results akin to the measurements obtained. The numerically derived data base would be used to generate design rules and correlations to guide the design of the film cooling configuration. Perhaps one of the more important extensions required to the present procedures would be the ability to cope with holes which were not normal to the surface. This could be achieved by the use of the embedded grid technique developed for

the leading edge film cooling computations. Indeed the essential features of the embedding procedure are not specific to the problem considered in the course of the present work and may be of use for other problems in fluid dynamics.

NOMENCLATURE

<u>Symbol</u>	<u>Meaning</u>
A^ϕ	finite difference coefficient
a	radial ordinate within hole
b^ϕ	coefficient of linearised source term
C	tracer gas mass fraction
C_{ax}	axial chord
C_p	pressure coefficient
c_1, c_2, c_μ	turbulence model constants
c_p	specific heat
C^ϕ	coefficient of linearised source term
D	hole diameter
DV_x, DV_y, DV_z	velocity correction coefficients
E	log-law wall function constant
F_i	body force per unit mass
f	weighting factor
f_m, f_p, f_y, f_τ	five-hole probe calibration functions
G	generation of turbulence kinetic energy
H	height of solution domain
h	total specific enthalpy (chapter 4)
h	heat transfer coefficient
J	jet to freestream velocity ratio
K	pressure gradient parameter
k	turbulence kinetic energy
k	thermal conductivity
L	turbulence length scale
l	length of hole

l_m	Prandtl mixing length
M	blowing rate $(\rho v)_s / (\rho v)_\infty$
M_n	Mach number
$\dot{m}$	mass flux
$\dot{m}^*$	mass flux residual on finite difference cell
$N_x, N_y, N_z, N_a, N_\beta, N_r, N_\theta$	number of grid nodes along subscripted ordinate
n	normal to solid surface
P	hole pitching
Pe	Peclet number
p	static pressure
p_T	total pressure
p_0, p_1, p_2, p_3, p_4	pressures registered by five-hole probe
p^*	estimated pressure
p'	pressure correction
Q	dynamic head
$\dot{q}''$	heat flux rate
R	radius of surface curvature
Ra	Rayleigh number
Re	Reynolds number
r	radial ordinate
S	slot height
St	Stanton number
s^ϕ	source term
T	temperature
T^+	normalised temperature
Tu	turbulence intensity $\frac{\sqrt{\frac{1}{3} u_i' u_i'}}{\sqrt{u_i u_i}}$
t	time

V_i	instantaneous velocity component
$V_x, V_y, V_z, V_a,$ V_β, V_r, V_θ	mean velocity component in direction of subscripted ordinate
V_τ	friction velocity parallel to surface
V^+	normalised velocity
V^*	apparent velocity $\sqrt{\frac{p_\tau - p_{wall}}{\frac{1}{2}\rho_\infty}}$
V_x^*, V_y^*, V_z^*	velocities arising from estimated pressure field p^*
X	traverse direction
X_D	downstream extent of solution domain
X_u	upstream extent of solution domain
x	axial ordinate
x_o	origin of turbulent boundary layer
Y	traverse direction
y	vertical ordinate
y^+	normalised distance from wall
y_p	normal distance from wall
Z	traverse direction
z	lateral ordinate
α	angle between slot/hole axis and surface
α_p	pitch angle of flow incident to five-hole probe
β	angular ordinate within hole
β_p	yaw angle of flow incident to five- hole probe
Γ	diffusion coefficient
γ	angle of hole axis in lateral direction

Δ	spacing of doublet singularities
$\Delta x, \Delta y, \Delta z$	cell dimensions
$\Delta\phi$	fractional change in quantity
δ	.995 boundary layer thickness
δ^*	boundary layer displacement thickness
δ_{ij}	Kronecker delta
ϵ	dissipation rate of turbulence
	kinetic energy
η	normalised scalar quantity, effectiveness
θ	angular ordinate around cylinder/ leading edge
K	log-law wall function constant
Λ	integral scale of turbulence
λ	relaxation factor
μ	molecular viscosity
μ^*	effective viscosity at a surface
μ_t	turbulent viscosity
ν	kinematic viscosity
ρ	density
σ^ϕ	molecular Prandtl/Schmidt number
σ_t^ϕ	turbulent Prandtl/Schmidt number
τ	time average integration period
τ_b	shear stress at solid boundary
Φ	velocity potential
ϕ	general variable
$\phi^{(2)}, \phi^{(1)}, \phi^{(2)}, \phi^{(3)}$	intermediate results obtained in the course of the ADI sequence
ψ	strength of doublet singularity

Subscript

Reference

a_w	adiabatic wall
B	solid boundary
f	with film
I	interface
i, j, k	components of Cartesian tensor notation
i_s	isothermal wall
i_w	impervious wall
j	jet
L	local
N, S, E, W, R, L, P	mesh nodes
n, s, e, w, r, l	cell faces
o	without film
t	turbulent
∞	freestream

Superscript

Meaning

$-$	time mean
$'$	fluctuating component
$\wedge$	area mean

REFERENCES

- Abramovich, G.N. (1963), The theory of turbulent jets. MIT Press.
- Adler, D. and Baron, A. (1979), Prediction of a three-dimensional circular jet in crossflow. AIAA Journal 17, 168.
- Afejuku, W.O, Hay, N., Lampard, D. (1980), The film cooling effectiveness of double rows of holes. J. Eng. Power, 102, 601.
- Ames, W.F. (1977), Numerical methods for partial differential equations. Nelson.
- Amsden, A.A. and Harlow, F.H. (1970), The SMAC method. Los Alamos Sci. Lab. Rep. LA-4370.
- Andreopoulos, J. (1980), Measurements in a pipe flow issuing perpendicularly into a cross-stream. ASME paper 80-WA/HT-24.
- Artt, D.W. (1970), An experimental investigation of film cooling, with particular reference to the injection region. Ph.D. thesis, Queen's Univ. Belfast.
- Baker, A.J. and Manhardt, P.D. (1979), A numerical viscous-inviscid interaction algorithm for prediction of near-field VSTOL jet-induced flowfields. NADC-77167-30.
- Bergeles, G. (1976), Three-dimensional discrete hole cooling processes an experimental and theoretical study. Ph.D. thesis, Univ. of London.
- Bowley, W.W. and Sucec, J. (1969), Trajectory and spreading of a turbulent jet in the presence of a crossflow of arbitrary

velocity distribution. ASME paper 69-GT-33.

Bradshaw, P. (1973), Effects of streamline curvature on turbulent flow. AGARDograph 169.

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

Burggraf, F. and Huffmeir, R.W. (1973), Film effectiveness and heat transfer coefficient downstream of a metered injection slot. ASME paper 73-HT-31.

Campbell, J.F. and Schetz, J.A. (1973), Analysis of the injection of a heated turbulent jet into a cross-flow. NASA TR R-413.

Carlson, L.W. and Talmor, E. (1968), Gaseous film cooling at various degrees of hot-gas acceleration and turbulence levels. Int. J. Heat and Mass Transfer, 11, 1695.

Castro, I.P. (1979), Numerical difficulties in the calculation of complex turbulent flows, Turbulent Shear Flows I, Eds. F. Durst et al, Springer Verlag, Berlin.

Chien, J.C. and Schetz, J.A. (1975), Numerical solution of the three-dimensional Navier-Stokes equations with applications to channel flows and a buoyant jet in a cross-flow. J. Appl. Mech., 42, E, 575.

Chieng, C.C. and Launder, B.E. (1980), On the calculation of turbulent heat transport downstream from an abrupt pipe expansion. Proc. Symp. Flow Heat Transfer in Recirc. Flows, Chicago, Illinois. Bk. No. GOO186, ASME.

Chorin, A.J. (1968), Numerical solution of the Navier-Stokes equations. Math. Comp., 22, 745.

Colladay, R.S. and Russell, L.M. (1975), Flow visualisation of discrete hole film cooling for gas turbine applications. NASA TM X-71766.

Crabb, D. (1979), Jets in crossflow. Ph.D. thesis, Univ. of London.

Crawford, M.E., Kays, W.M., Moffat, R.J. (1980), Full coverage film cooling. Part II Heat transfer data and numerical simulation. ASME paper 80-GT-44.

Dring, R.P., Blair, M.F. Joslyn, H.D. (1978), Research on centrifugal effects on turbine rotor blade film cooling. AFAPL TR-78-63.

Dvorak, F.A. (1973), Calculation of turbulent boundary layers and wall jets over curved surfaces. AIAA Journal 11, 517.

Elovic, E. and Koffel, W.K. (1979), Some considerations in the thermal design of turbine airfoil cooling systems. ASME paper 79-GT/Isr-20.

Eriksen, V.L. (1971), Film cooling effectiveness and heat transfer with injection through holes. NASA CR-72991.

Eriksen, V.L., Eckert, E.R.G., Goldstein, R.J. (1971), A model for analysis of the temperature field downstream of a heated jet injected into an isothermal cross-flow at an angle of 90° . NASA CR-72990.

Folayan, C.O. and Whitelaw, J.H. (1976), The effectiveness of two-dimensional film cooling over curved surfaces. ASME paper 76-HT-31.

Foss, J.F. (1980), Interaction region phenomena for the jet in a cross-flow problem. SFB 80/E/161, Univ. of Karlsruhe.

Foster, R.C. (1973), Experimental study of boundary layer and heat transfer in the region of flow downstream of normal injection slots. Ph.D. Thesis Univ. of Texas.

Foster, N.W. and Lampard, D. (1974), Some experiments on the film cooling effectiveness of a single row of holes. Univ. of Nottingham Mech. Eng. Dept. Report.

Foster, N.W. and Lampard, D. (1979), The flow and film cooling effectiveness following injection through a row of holes. ASME paper 79-GT/Isr - 6.

Fullagar, K.P.L. (1973), The design of air cooled turbine rotor blades. Presented at Symposium on Design and Calculation of Constructions subject to high temperature, Univ. of Tech., Delft, Netherlands.

Gauntner, J.W. (1977), Effects of film ejection angle on turbine vane cooling. NASA TP 1095.

Gibson, M.M. (1978), An algebraic stress and heat-flux model for turbulent shear flow with streamline curvature. Int. J. Heat Mass Transfer, 21, 1609.

Glushko, G. (1965), Turbulent boundary layer on a flat plate in an incompressible fluid. Izv. Acad. Nauk SSSR Mekh No. 4, 13.

Goldstein, R.J., Eckert, E.R.G., Burggraf, F. (1974), Effects of hole geometry and density on three-dimensional film cooling. Int. J. Heat and Mass Transfer, 17, 595.

Goldstein, R.J., Eckert, E.R.G., Eriksen, V.L., Ramsey, J.W. (1969), Film cooling following injection through inclined circular tubes. NASA CR-72612.

Goldstein, R.J., Eckert, E.R.G., Ramsey, J.W. (1968), Film Cooling with injection through a circular hole. NASA CR-54606.

Goldstein, R.J., Eriksen, V.L., Ramsey, J.W. (1978), Flow and temperature fields following injection of a jet normal to a cross-stream. Proc. 6th Int. Heat Transfer Conf., Toronto.

Goldstein, R.J. and Haji-Sheikh, A. (1967), Prediction of film cooling effectiveness. Proc. Japan Soc. Mech. Engr. Semi-Intern. Symp., 213.

Gosman, A.D., Pun, W.M., Runchal, A.K., Spalding, D.B., Wolfshtein, M. (1969), Heat and mass transfer in recirculating flows. Academic Press.

Gosman, A.D. and Pun, W.M. (1974), Lecture notes for a course entitled 'Calculation of Recirculating Flows.' Imperial College, London.

Halls, G.A. (1971), Nozzle guide vane cooling - the - state - of - the - art. In High Temperature Turbines, AGARD CP-73, Paper 25.

Hanjalić, K. and Launder, B.E. (1972), A Reynolds stress model of turbulence and its application to asymmetric shear flows. J. Fluid Mech. 52, 609.

Hanus, G.J. and L'Ecuyer, M.R. (1976), Turbine Vane leading edge gas film cooling with spanwise angled coolant holes. AIAA paper 76-43.

Harlow, F.H. and Nakayama, P.I. (1968), Transport of turbulence energy decay rate. Los Alamos Sci. Lab. Univ. California Rep. LA-3854.

Harlow, F.H. and Welch, J.E. (1965), Numerical calculation of time-dependant viscous incompressible flow of fluid with free-surface. Phys. Fluids, 8, 2182.

Hartnett, J.P., Birkebak, R.C., Eckert, E.R.G. (1961a), Velocity distributions, temperature distributions, effectiveness and heat transfer for air injected through a tangential slot into a turbulent boundary layer. J. Heat Transfer, 83, C, 293.

Hartnett, J.P., Birkebak, R.C., Eckert, E.R.G. (1961b), Velocity distributions, temperature distributions, effectiveness and heat transfer in cooling of a surface with a pressure gradient. International Developments in Heat Transfer, Part IV, Section A, 682. ASME New York.

Hartsel, J.E. (1970), An experimental investigation of the effects of massive film cooling on the aerodynamics of a turbine aerofoil. Air Force Aero. Prop. Lab. Rep. AFAPL-TR-70-97.

Hartsel, J.E. (1972), Prediction of effects of mass-transfer cooling on the blade-row efficiency of turbine aerofoils. AIAA paper 72-11.

Herring, H.J. (1974), A method of predicting the behaviour of a turbulent boundary layer with discrete transpiration jets. ASME paper 74-GT-48.

Hinze, J.O. (1975), Turbulence. McGraw - Hill.

Hwang, S.S. and Pletcher, R.H. (1978), Prediction of buoyant turbulent jets and plumes in a cross-flow. Proc. 6th Int. Heat Transfer Conf., Toronto.

Irving, H.P.A.H. and Arnot-Smith, P. (1975), Prediction of the effect of streamline curvature on turbulence. Phys. Fluids, 18, 624.

Ito, S., Goldstein, R.J., Eckert, E.R.G. (1978), Film cooling of a gas turbine blade. J. Eng. for Power, 100, 476.

Jabbari, M.Y. (1973), Film cooling and heat transfer with air injection through a staggered row of holes into an accelerating flow. Ph. D. Thesis, Univ. of Minnesota.

Jayatillaka, C.L.V. (1969), The influence of Prandtl number and surface roughness on the resistance of the laminar sublayer to momentum and heat transfer. In Progress in Heat and Mass Transfer,

1 Ed. U. Grigull and E. Hahne, Pergamon Press.

Jones, W.P. (1976), A computer programme for solving three-dimensional flow problems. Rolls-Royce Internal Report.

Jones, W.P. and Launder, B.E. (1972), The prediction of laminarisation with a two-equation model of turbulence. Int. J. Heat Mass Transfer, 15, 301.

Jones, W.P. and McGuirk, J.J. (1979), Computation of a round turbulent jet discharging into a confined cross-flow. Turbulent Shear Flows 2, Springer - Verlag, Heidelberg.

Kacker, S.C., Pai, B.R., Whitelaw, J.H. (1968), The prediction of wall-jet flows with particular reference to film cooling. Imp. Coll. Mech. Eng. Dept. Rep. TWF/TN/43.

Kadotani, K. and Goldstein, R.J. (1977a), On the nature of jets entering a turbulent flow. Part A : Jet mainstream interaction. Paper 05-A ASME/JSME Joint Gas Turbine Congress, Tokyo.

Kadotani, K. and Goldstein, R.J. (1977b), On the nature of jets entering a turbulent flow. Part B : Film cooling performance. Paper 05-B ASME/JSME Joint Gas Turbine Congress, Tokyo.

Kadotani, K. and Goldstein, R.J. (1979), Effect of mainstream variables on jets issuing from a row of inclined round holes. J. Eng. for Power, 101, 298.

Keffer, J.F. and Baines, W.D. (1963), The round turbulent jet in a crosswind. J. Fluid Mech. 15, 481.

Klebanoff, P.S. and Diehl, Z.W. (1952), Some features of artificially thickened fully developed turbulent boundary layers with zero pressure gradient. NACA Rep 1110.

Kolmogorov, A.N. (1942), Equations of turbulent motion of an incompressible turbulent fluid. Izv. Akad. Nauk SSSR Ser Phys. VI, No. 1-2, 56.

Kreskovsky, J.F., Briley, W.E., McDonald, H. (1980), Computation of discrete hole film cooling flow using the Navier-Stokes equations. AFOSR-TR-79-0935.

Kruse, H. (1974), Film cooling measurements. DFVLR Internal Report 352-74/9.

Kutateladze, S.S. and Leont'ev, A.I. (1963), Film cooling with a turbulent gaseous boundary layer. Thermal Phys. High Temp., 1, 281.

Lamb, H. (1932), Hydrodynamics, 6th ed. Cambridge University Press.

Lander, R.D., Fish, R.W., Suo, M. (1972), External heat transfer distribution on film cooled turbine vanes. J. Aircraft, 9, 707.

Launder, B.E., Morse, A.P., Rodi, W., Spalding, D.B. (1972), The prediction of free shear flows - a comparison of six turbulence models. Paper 11 in NASA SP-321.

Launder, B.E., Reece, G.J., Rodi, W. (1975), Progress in the development of a Reynolds stress turbulence closure. J. Fluid Mech, 86, 537.

Lauder, B.E. and Spalding, D.B. (1972), Mathematical models of turbulence. Academic Press.

Lauder, B.E. and Spalding, D.B. (1974), The numerical computation of turbulent flows. Computer Methods in Applied Mechanics and Engineering, 3 , 269.

Lauder, B.E. and York, J. (1974), Discrete - hole cooling in the presence of free-stream turbulence and favourable pressure gradient. Int. J. Heat and Mass Transfer, 17, 1403.

Leonard, B.P. (1979), A stable and accurate convective modelling procedure based on quadratic upstream interpolation. Computer Methods in App. Mech. and Eng., 19, 59.

Librizzi, J. and Cresci, R.J. (1964), Transpiration cooling of a turbulent boundary layer in an axisymmetric nozzle. AIAA J., 2, 617.

Liess, C. (1974), Experimental investigation of film cooling with ejection from a row of holes for the application to gas turbine blades. ASME paper 74-GT-5.

Liess, C. and Carnel, J. (1971), Application of film cooling to gas-turbine blades. In High Temperature Turbines, AGARD CP-73, Paper 23.

Louis, J.F., Demirjian, A.M., Goulios, G.N., Topping, R.F., Wiedhopf, J.M. (1974), Short duration studies of turbine heat transfer and film cooling effectiveness. ASME paper 74-GT-131.

Luckey, D.W. and L'Ecuyer, M.R. (1981), Stagnation region gas film cooling : Spanwise angled injection from multiple rows of holes. NASA CR-165333.

Luckey, D.W., Winstanley, D.K., Hanus, G.J., L'Ecuyer, M.R. (1977), Stagnation region gas film cooling for turbine blade leading edge applications. J. Aircraft, 14, 494.

McGuirk, J.J., and Rodi, W. (1978), A depth-averaged mathematical model for the near-field of side-discharges into open channel flow. J. Fluid Mech., 86, 761.

McGuirk, J.J., Taylor, A.M.K.P., Whitelaw, J.H. (1981), The assessment of numerical diffusion in upwind-difference calculations of turbulent recirculating flows. Paper presented at : Third Symposium on Turbulent Shear Flows, Univ. of California.

Marek, C.J. and Tacina, R.R. (1975), Effects of freestream turbulence on film cooling. NASA TN D-7958.

Matthews, L. (1972), Performance of some two dimensional film cooling geometries. Ph. D. thesis Univ. of London.

Mayle, R.E. and Kopper, F.C. (1976), Adiabatic wall effectiveness of a turbulent boundary layer with slot ejection. J. Heat Transfer, 98, C, 240.

Mayle, R.E., Kopper, F.C., Blair, M.F., Bailey, D.A. (1977), Effect of streamline curvature on film cooling. J. Eng. for Power, 99, A, 77.

Metzger, D.E., Carper, J.H., Warren, J.M. (1972), Predicted film cooling near flush slots - comparison with experiment. J. Aircraft, 9, 857.

Metzger, D. E. and Fletcher, D.D. (1969), Surface heat transfer immediately downstream of flush non-tangential injection holes and slots. AIAA paper 69-523.

Minty, A.G. (1973), An investigation of the effect of slit angle film cooling performance and consideration of multiple slit arrangements for turbine blade design purposes. Ph. D. thesis, UWIST.

NASA (1969), Analysis of a jet in a subsonic crosswind. Special report SP 218.

Nicolas, J. and LeMeur, A. (1974), Curvature effects on a turbine blade cooling film. ASME paper 74-GT-156.

Papell, S.S. (1960), Effect on gaseous film cooling of coolant injection through angled slots and normal holes. NASA TN D - 299.

Patankar, S.V., Basu, D.K., Alpay. S.A. (1977), Predictions of the three-dimensional velocity field of a deflected turbulent jet. J. Fluids Eng., 99, I, 758

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.

Peaceman, D.W. and Rachford, H.H. (1955), The numerical solution of parabolic and elliptic differential equations. J. Soc. Indust. Appl. Math., 3, 28.

Pedersen, D.R. (1972), Effect of density ratio on film cooling effectiveness for injection through a row of holes and for a porous slot. Ph. D. thesis Univ. of Minnesota.

Pope, S.B. and Whitelaw, J.H. (1976), The calculation of near-wake flows. J. Fluid Mech., 73, 9.

Prandtl, L. (1925), Bericht uber Untersuchungen zur ausgebildeten Turbulenz. ZAMM, 5, 136.

Priddin, C.H. (1975), The turbulent boundary layer on curved, porous walls. Ph. D. thesis Univ. of London.

Priddin, C.H. (1979), Private Communication.

Raithby, G.D. (1976a), A critical evaluation of upstream differencing applied to problems involving fluid flow. Comp. Meth. Appl. Mech. and Eng., 9, 75.

Raithby, G.D. (1976b), Skew upstream differencing schemes for problems involving fluid flow. Comp. Meth. Appl. Mech. Eng., 9, 153.

Ramsey, J.W. and Goldstein, R.J. (1971), Interaction of a heated jet with a deflecting stream. ASME paper 71-HT-2.

Rastogi, A.K. and Whitelaw, J.H. (1970), A procedure for predicting the influence of longitudinal curvature on boundary layer flows. Imperial College Mech. Eng. Dept. Rep. BL/TN/A/37.

Rastogi, A.K. and Whitelaw, J.H. (1972), Film cooling from a row of holes located along a cylinder in cross-flow. Imperial College Mech. Eng. Dept. Rep. ET/TN/A/17.

Robertson, J.M. (1965), Hydrodynamics in Theory and Application.

Rodi, W. (1976), A new algebraic relation for calculating Reynolds stresses. ZAMM, 56, 219.

Rodi, W. (1981), Progress in turbulence modelling for incompressible flows. AIAA paper 81 - 0045.

Rodi, W. and Srivatsa, S.K. (1980), A locally elliptic calculation procedure for three-dimensional flows and its application to a jet in cross-flow. Comp. Meth. Appl. Mech. and Eng., 23, 67.

Rubesin, M.W. (1951), The effect of arbitrary surface temperature variation along a flat plate on the convective heat transfer in an incompressible turbulent boundary layer. NACA TN-2345.

Russell, L.M. (1979), Flow visualisation of discrete - hole film cooling with spanwise injection over a cylinder. NASA TP - 1491.

Sasaki, M., Takahara, K., Sakata, K., Kumagai, T. (1976), Study on film cooling of turbine blades, first report : Experiments on film cooling with injection through holes near a leading edge. Bull. J.S.M.E., 19, 1344.

Sawyer, R.A. (1963), Two-dimensional reattaching jet flows including the effects of curvature on entrainment. J. Fluid Mech, 17, 481.

Scesa, S. (1954), Effect of normal blowing on flat plate heat transfer. Ph. D. thesis, Univ. of California.

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

Sellers, J.P., Jr., (1963), Gaseous film cooling with multiple injection stations. AIAA J., 1, 2154.

Sivasegaram, S. (1967), Film cooling slots : the importance of lip thickness and injection angle. M.Sc. thesis, Univ. of London.

Smith, M.R., Jones, T.V., Schultz, D.L. (1974), Film cooling effectiveness from rows of holes under simulated gas turbine conditions. Aero. Res. Council report ARC 34738 HMT 333.

Spalding, D.B. (1965), Prediction of adiabatic wall temperatures in film-cooling systems. AIAA J., 3, 965..

Spalding, D.B. and Patankar, S.V. (1967), Heat and mass transfer in boundary layers. Intertext books, London.

Sterland, P.R. and Hollingsworth, M.A. (1975), An experimental study of multiple jets directed normally to a cross-flow. I.Mech.E. J. Mech. Eng. Sci, 17, 117.

Stollery, J.L. and El - Ehwany, A.A.M. (1965), A note on the use of a boundary layer model for correlating film cooling data. Int. J. Heat Mass Transfer, 8, 55.

Sturgess, G.J. and Lenertz, J.E. (1977), Account of mainstream turbulence for predicting film cooling effectiveness in gas turbine combustors. ASME paper 77-HT-10.

Taneda, S. (1956), J. Phys. Soc. Japan, 11, 302.

Taylor, A.M.K.P. and Whitelaw, J.H. (1975), The effectiveness of discrete hole arrangements for cooling the leading edge of turbine blades, Imperial College Mech. Eng. Dept. Rep.

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

Tribus, M. and Klein, J. (1953), Forced convection from non-isothermal surfaces. In Proc Symposium on Heat Transfer, Univ. Michigan, Ann Arbor., 211.

De Vahl Davies, G. and Mallinson, G.D. (1972), False diffusion in numerical fluid mechanics. Univ. of New South Wales, School of Mech. and Ind. Eng. Rept. 1972/FMT/1.

Vasilic - Melling, D. (1976), Three-dimensional turbulent flow past rectangular bluff bodies. Ph. D. thesis, Univ. of London.

Wang, C.R., Papell, S.S., Graham, R.W. (1981), Analysis for predicting adiabatic wall temperatures with single hole coolant ejection into a low speed cross-flow. ASME paper 81-GT-91.

Warren, C.H.E. (1960), An experimental investigation of the effect of ejecting a coolant gas at the nose of a bluff body. J. Fluid Mech., 8, 400.

Warren, J.M. and Metzger, D.E. (1972), Heat transfer with film cooling in the presence of a laminarizing mainstream. ASME paper 72-HT-11.

Weighardt, K. (1946), Hot-air discharge for de-icing. A.A.F. Translation, Rep. No. F-TS-919-Re, Wright Field.

Wolfshtein, M. (1968), Numerical smearing in one sided difference approximations to the equations of non-viscous flow. Imperial College, Mech. Eng. Rep HTS/68/2.

Wynnanski, I. and Newmann, B.G. (1968), The reattachment of an inclined two-dimensional jet to a flat surface in streaming flow. C.A.S.I. Trans., 1, 3.

APPENDIX I

POTENTIAL FLOW ANALYSIS OF CYLINDER

IN WINDTUNNEL

An analytic solution of the potential flow around a circular cylinder mounted centrally in a windtunnel has been obtained using superposition. Many texts have described how the superposition of a uniform flow and a doublet singularity with the source facing upstream may be used to obtain the potential flow solution for a circular cylinder in an infinite crossflow. Using the body centred co-ordinate system shown in figure A1.1 the potential functions are :-

$$\begin{aligned}\Phi_{cyl} &= \Phi_{\Rightarrow} + \Phi_{\odot} \\ &= V_{\infty} x + \frac{\psi x}{x^2 + y^2}\end{aligned}\quad (A1 - 1)$$

where ψ is the doublet strength.

The radius of the cylinder so defined is given by

$$R = \sqrt{\frac{\psi}{V_{\infty}}}\quad (A1 - 2)$$

With the addition of a second doublet singularity of the same strength and orientation at some point on the y -ordinate, the streamline bisecting the y - axis between the two doublets becomes straight. This solution may be interpreted as describing the potential flow about a circular cylinder near a flat wall defined by the bisecting streamline. In this case the second doublet, which lies within the wall, is termed an image singularity.

For the arrangement shown in figure A1.2 the potential function becomes :

$$\begin{aligned}\Phi &= \Phi_{\infty} + \Phi_{00} + \Phi_{10} \\ &= V_{\infty} x + \frac{\psi_{00} x}{x^2 + y^2} + \frac{\psi_{10} x}{x^2 + (y - \Delta)^2} \quad (A1-3)\end{aligned}$$

Where the subscripts define the location and image order of the doublets (see later)

$$\text{and} \quad \psi_{00} = \psi_{10} = \psi$$

As indicated by Lamb (1932), when the doublet is located approximately twice the cylinder radius from the wall, the cylinder becomes significantly elongated in the flow direction. To overcome this distortion, which rapidly becomes worse as the doublet is brought nearer the wall, corrective terms must be introduced in the form of additional images located within the cylinders. Robertson (1965) describes the arrangement of these images. Referring to figure A1.2 we have :

$$\Phi = \Phi_{\infty} + \Phi_{00} + \Phi_{10} + \Phi_{01} + \Phi_{11} + \dots \quad (A1 - 4)$$

$$\begin{aligned}&= V_{\infty} x + \frac{\psi_{00} x}{x^2 + y^2} + \frac{\psi_{10} x}{x^2 + (y - \Delta)^2} + \frac{\psi_{01} x}{x^2 + (y - \Delta_{01})^2} \\ &\quad + \frac{\psi_{11} x}{x^2 + (y - \Delta + \Delta_{11})^2} \quad (A1 - 5)\end{aligned}$$

Where, as described by Robertson (1965),

$$\psi_{01} = -\psi_{10} \left(\frac{R}{\Delta} \right)^2; \quad \psi_{11} = -\psi_{00} \left(\frac{R}{\Delta} \right)^2 \quad (\text{A1 - 6})$$

$$\text{and } \Delta_{01} = \Delta_{11} = \frac{R^2}{\Delta}$$

The first subscript has been used to specify the general location of the singularity, i.e. 0 if it is in the upper cylinder and 1 if it is in the lower. The second subscript implies the order of the image, i.e. zero or first. In fact the image doublets ψ_{01} and ψ_{11} set up further images ψ_{12} and ψ_{02} within the cylinders which, in turn, give rise to yet more images leading to an infinite series. For the configuration of interest, however, these are negligible from images of the second order, ψ_{12} and ψ_{02} , and above.

As shown in figure A1.2 the potential flow described by equations A1-5 and A1-6 gives rise to a curved stagnation streamline, whereas, to model the cylinder in the windtunnel, the stagnation streamline must be straight. The required condition is obtained by generalising the above analysis to an infinite row of equi-spaced doublets. Referring to figure A1-3 we have :

$$\Phi = V_{\infty}x + \sum_{m=-\infty}^{m=+\infty} \sum_{n=-\infty}^{n=+\infty} \left[\frac{\psi_{n1}x}{x^2 + (y - \Delta_n - m\Delta)^2} \right] \quad (\text{A1 - 7})$$

$$\left. \begin{aligned} \text{Where } \psi_{n1} &= -\psi_{00} \left(\frac{R}{n\Delta} \right)^2 \\ \Delta_{n1} &= \frac{R^2}{n\Delta} \end{aligned} \right\} n \neq 0$$

$$\left. \begin{aligned} \psi_{n1} &= \psi_{00} \\ \Delta_{n1} &= 0 \end{aligned} \right\} n=0$$

In Eqn. A1-7 the index m refers to the location of the doublet in the row and the index n refers to the images of the adjacent doublets within the cylinder, i.e. $n = 1$ refers to the image of the $m+1$ th doublet within the m th cylinder of the row.

The velocity components which result are given by :

$$V_x = \frac{\partial \Phi}{\partial x} = V_\infty + \sum_{m=-\infty}^{\infty} \sum_{n=-\infty}^{\infty} \psi_{n1} \left\{ \frac{(y - \Delta_{n1} - m\Delta)^2 - x^2}{[x^2 + (y - \Delta_{n1} - m\Delta)^2]^2} \right\} \quad (A1-8)$$

$$V_y = \frac{\partial \Phi}{\partial y} = \sum_{m=-\infty}^{\infty} \sum_{n=-\infty}^{\infty} \left\{ \frac{-2\psi_{n1}x(y - \Delta_{n1} - m\Delta)}{[x^2 + (y - \Delta_{n1} - m\Delta)^2]^2} \right\} \quad (A1-9)$$

The velocity field about the cylinder has been determined by a computer program written specifically for the evaluation of Eqns. (A1 - 8) and (A1 - 9), the summation including all terms of the series greater than 10^{-4} of V_∞ .

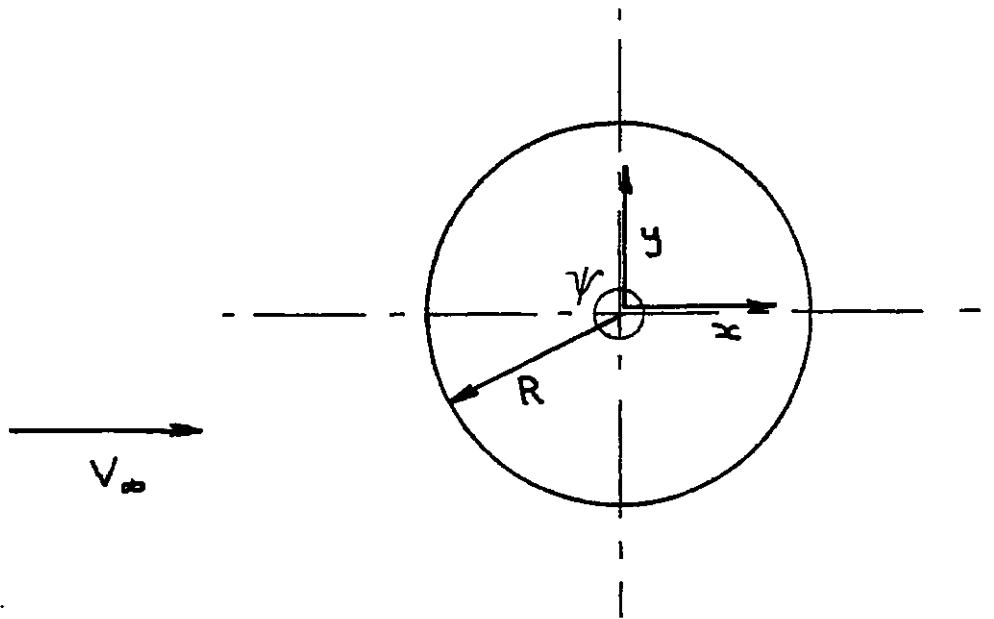


Fig. A1.1 Cylinder in an infinite crossflow

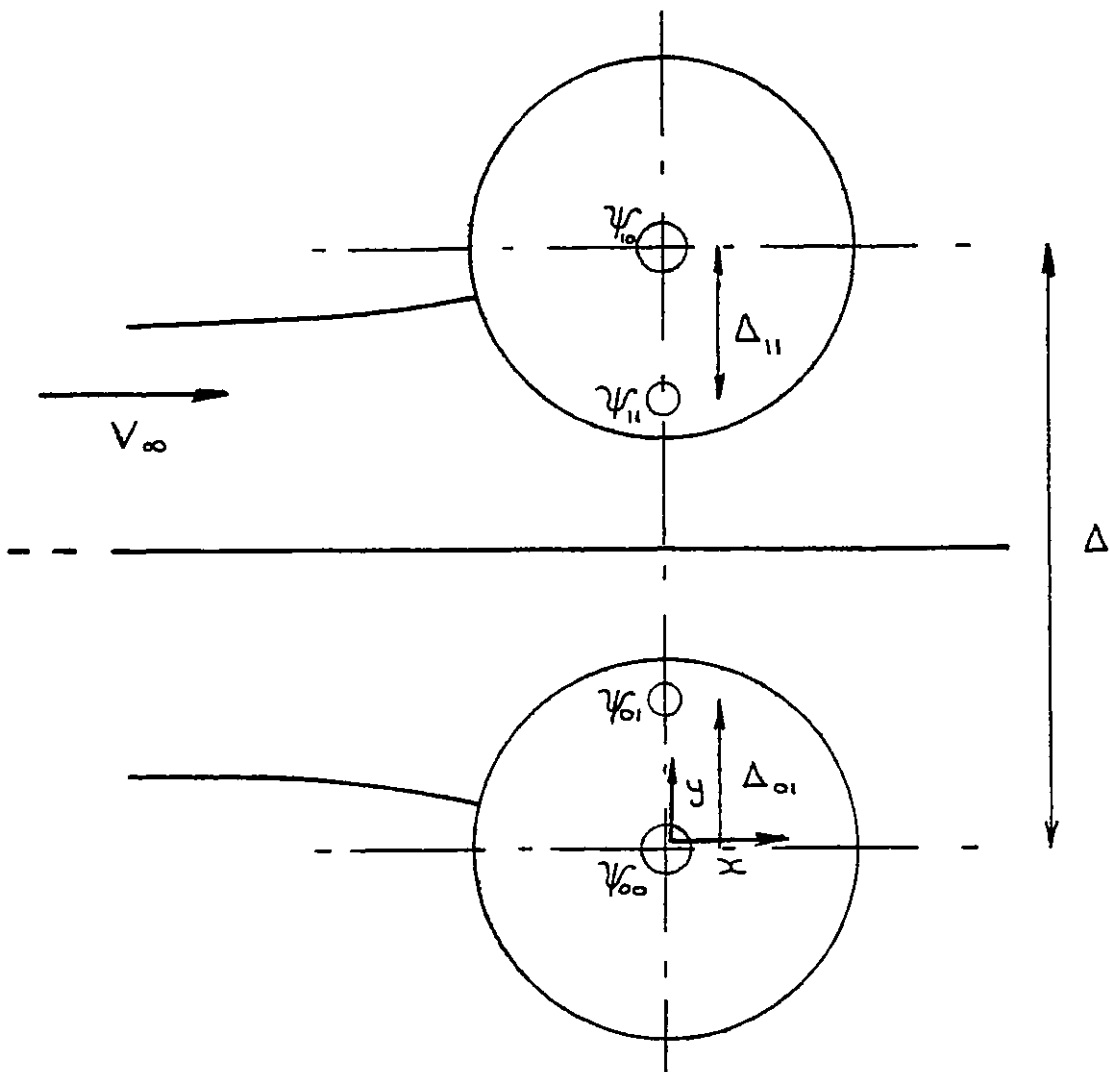


Fig. A1.2 Doublet arrangement to represent a cylinder adjacent to a wall

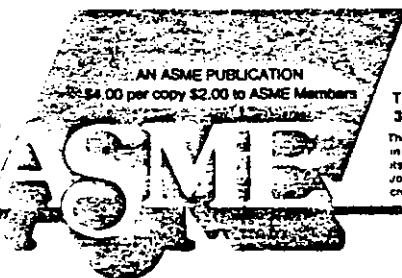
APPENDIX II

ASME PAPER 80 - WA/HT - 26.

The prediction of the flow and heat transfer
in the vicinity of a jet in crossflow.

by A.J. White

Presented at the ASME Winter Annual Meeting,
November, 1980, Chicago, Illinois.



80-WA/HT-26

THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS
345 E 47 St., New York, N.Y. 10017

The Society shall not be responsible for statements or opinions advanced in papers or in discussion at meetings of the Society or of its Divisions or Sections, or printed in its publications. Discussion is invited only if the paper is published in an ASME Journal or Proceedings. Release for general publication upon presentation. Full credit should be given to ASME, the Technical Division, and the author(s).

The Prediction of the Flow and Heat Transfer in the Vicinity of a Jet in Crossflow

A. J. White

Senior Blade Cooling Research Engineer,
Rolls Royce Ltd.,
Derby, England

A procedure for solving the fully-elliptic time averaged Navier-Stokes equations describing turbulent flow is applied to the calculation of the flow and heat transfer arising from the injection of fluid through a wall into a cross-flowing mainstream. The solutions include the use of the numerical procedure to calculate the distortion of the flow within the injection hole, iteration being carried out between this calculation and that of the external flow in order to obtain converged solutions of flow-field. Predictions are presented for injection from a single normal hole at jet to mainstream mass flux ratios between 0.24 and 2.3. At these conditions the distortion of the flow within the hole is found to have a significant effect on the predicted results. The agreement between the predicted results and experimental data is encouraging.

NOMENCLATURE

C_F, C_1, C_2	turbulence model constants
D	hole diameter
h	heat transfer coefficient
J	momentum ratio, $(\rho_T V_T^2 / \rho_\infty U_\infty^2)$
K	turbulence kinetic energy, $\frac{1}{2} \overline{u_i' u_i'}$
L	turbulence length scale
M	blowing rate, $(\rho_T V_T / \rho_\infty U_\infty)$
p	static pressure
Pr	Prandtl number
r	radial co-ordinate
T	temperature
U	x -direction velocity component
V	y -direction velocity component
W	θ, z -direction velocity component
u', v', w'	fluctuating velocity components
x	axial co-ordinate
y	vertical co-ordinate

z	transverse co-ordinate
X, Y, X	calculation domain boundaries
δ_{ij}	Kronecker delta
ϵ	dissipation rate of turbulence kinetic energy
η	non-dimensional temperature, effectiveness: $(T - T_\infty) / (T_j - T_\infty)$
θ	tangential co-ordinate
μ	laminar viscosity
ρ	density
Gr, Gr_x, Gr_z	turbulent Prandtl numbers for T, X and z

SUBSCRIPTS

C	cylindrical co-ordinate system components
D	downstream
i	with injection
J	jet
o	without injection
U	upstream
∞	freestream

Contributed by the Heat Transfer Division of The American Society of Mechanical Engineers for presentation at the Winter Annual Meeting, November 16-21, 1980, Chicago, Illinois. Manuscript received at ASME Headquarters July 12, 1980.

Copies will be available until August 1, 1981.

INTRODUCTION

Problem Considered

Film cooling has for many years been employed in cooling the rotor blades and stators of gas turbine engines. Air bled from the compressor is injected through rows of holes in the surface of components exposed to high gas temperatures. Such a method of cooling allows operation of engines at high temperatures, improving the theoretical efficiency of the engine, however there are drawbacks. Although cooling by air bled from the compressor is the only currently practicable method for aircraft gas turbines it involves an inevitable reduction in the actual efficiency gain. Further losses arise in the case of film cooling due to the mixing of the injected flow with that over the surface of the component. Film cooling can also give rise to large thermal gradients in the material of the cooled component leading to large stress concentrations with the possibility of cracking the component. Clearly it is important to minimise the air bled from the compressor for cooling purposes, whilst ensuring that the film cooling maximises thermal protection with minimal temperature gradients, and aerodynamic loss.

The work presented in this paper comprises a theoretical study of the flow from a single hole exhausting perpendicularly from a wall into a cross-flowing mainstream. The theoretical method employed involves the direct solution of the fully elliptic time averaged Navier-Stokes equations describing turbulent flow by means of finite differences with the Reynolds stresses being modelled by the $K-\epsilon$ turbulence model (Lauder and Spalding (1)). The numerical solution therefore gives the full three dimensional velocity and pressure fields around the jet, together with the temperature field obtained from the solution of an equation for the transport of heat. In view of this it is appropriate that the theoretical solutions be compared with experimental data which include measurements of both aerodynamic and heat transfer parameters. Unfortunately little aerodynamic data exist for rows of holes, most investigations having measured film cooling effectiveness alone. Attention has therefore been confined to the flow from single holes, for which both aerodynamic and heat transfer measurements are available.

The flowfield set up by a jet in crossflow is shown in Fig. 1. Upstream of the jet the mainstream is retarded and divides to pass on either side of the jet, which in turn is bent by the mainstream by an amount dependent on the relative momenta of the jet and mainstream flows. Downstream, the flow is dominated by two contra-rotating streamwise vortices which, where the jet lifts off the surface, are enhanced by entrainment into the underside of the jet, and which may persist for a considerable distance downstream.

Previous Work

Previous studies on the application of similar theoretical methods to the jet in crossflow include those of Bergeles (2), Chien and Schetz (3), Patankar et al. (4), and Crabb (5). In a recent paper Jones and McGuirk (6) used the same numerical method as that employed in the present work to study high velocity ratio confined jets.

Bergeles (2) employed a semi-elliptic calculation method and $K-\epsilon$ turbulence modelling to calculate the flow from a single hole and rows of holes. The semi-elliptic scheme is an extension

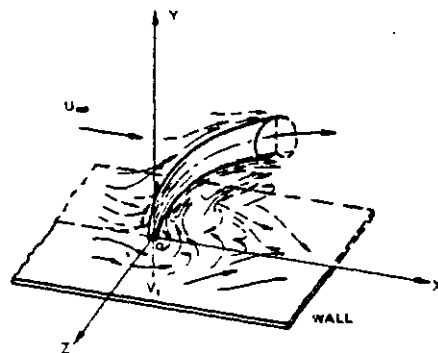


Fig. 1 Flowfield around a jet in crossflow

of three-dimensional parabolic calculation techniques. The procedure allows upstream effects in the pressure field, but cannot cope with flow reversal in the streamwise direction. In its application to jets in crossflow, the semi-elliptic method is consequently restricted to low jet to mainstream velocity ratios ($V_j/U_\infty < 0.1$ for normal injection).

Chien and Schetz (3) used a three dimensional fully elliptic numerical method with velocity-vorticity formulation to calculate a turbulent buoyant jet in crossflow. The calculations were completed on a $24 \times 22 \times 22$ grid and employed a constant Prandtl eddy viscosity model. Good agreement was obtained with jet trajectory data, however a CPU requirement of several hours on an IBM 370-158 is quoted.

The studies of Patankar et al. (4) and Crabb (5) both applied direct solution of the Navier-Stokes equations by finite difference techniques and $K-\epsilon$ turbulence modelling to the jet in crossflow. In the work of Patankar et al. (4) the finite difference grid was restricted to $15 \times 15 \times 10$ nodes (in the X, Y and Z directions respectively) by the limited storage capacity of the available computing facilities. The effect of this limitation was reduced by using computational experiment to determine the minimum acceptable size of the calculation domain. In order to determine the grid dependence of the solutions a calculation was completed on a $12 \times 12 \times 8$ grid, the results of which were within 5 percent of the finer grid calculation. In comparisons with experiment satisfactory agreement was obtained with jet trajectory data, and good agreement with data giving the decay of the maximum jet velocity. Reasonable agreement was obtained with the velocity profile measurements of Ramsey and Goldstein (7).

In the work of Crabb (5) a finite difference grid equivalent to $24 \times 14 \times 14$ nodes (in the X, Y and Z directions) was employed. The calculation domain boundaries were determined by the dimensions of the wind tunnel employed by Crabb to obtain the data with which the predictions were compared. As a result the calculation grid was relatively coarse and this is reflected in the solutions obtained. The measurements included in this study were obtained by Laser Doppler anemometry and hence provide accurate

data with which prediction methods may be compared.

Outline of the Paper

The aim of the present work is to improve the accuracy of the solutions obtained by these methods for the jet in crossflow. One of the major limitations of this work arises from the limited definition attainable with the finite difference grid. Consequently a number of calculations are completed in order to study the grid dependence of the solution, and computational experiment is employed to determine the minimum acceptable calculation domain size. A physical feature of the flow, which has not been included in the previous work, is the distortion of the flow within the hole arising from the pressure field set up by the jet mainstream interaction. This is included in the present study, a separate calculation being employed to determine the development of the flow within the hole. A brief description of the numerical method employed, and details of the computational arrangements of the present calculations are presented in the next sections. These are followed by a discussion of the results and their comparison with previous work and available measured data.

BASIC THEORY

The flow set up by a jet discharging normally into a crossflowing stream is a complex three-dimensional field involving upstream influence, turbulent mixing, large cross stream gradients and, at sufficiently high jet velocities, regions of reverse flow. The equations governing this type of flow are the full three-dimensional equations of motion for turbulent fluid flow. Since we are also interested in the temperature field we also solve an equation for the transport of heat. In cartesian tensor notation the appropriate time averaged form of these equations are:-

$$\frac{\partial \bar{\rho} \bar{U}_i}{\partial x_i} = 0 \quad (1)$$

$$\bar{\rho} \bar{U}_i \frac{\partial \bar{U}_j}{\partial x_i} = -\frac{\partial \bar{p}}{\partial x_j} + \frac{\partial}{\partial x_i} \left[\mu \left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) - \bar{\rho} \bar{u}_i \bar{u}_j \right] \quad (2)$$

$$\bar{\rho} \bar{U}_i \frac{\partial \bar{T}}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\frac{\mu}{Pr} \frac{\partial \bar{T}}{\partial x_i} - \bar{\rho} \bar{u}_i \bar{T} \right] \quad (3)$$

The Reynolds stresses $\bar{\rho} \bar{u}_i \bar{u}_j$ and turbulent temperature fluctuation correlation $\bar{\rho} \bar{u}_i \bar{T}$ require modelling. In the present work the two-equation $K - \epsilon$ turbulence model has been employed. In this model Reynolds stresses are linearly related to the mean rate of strain by a turbulent viscosity, μ_t :

$$\bar{\rho} \bar{u}_i \bar{u}_j = \frac{2}{3} \bar{\rho} K \delta_{ij} - \mu_t \left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) \quad (4)$$

where $\mu_t = \bar{\rho} C_\mu K / \epsilon$; $K = \frac{1}{2} \bar{u}_i \bar{u}_i$

$$\text{and } \bar{\rho} \bar{u}_i \bar{T} = -\frac{\mu_t}{\sigma_T} \frac{\partial \bar{T}}{\partial x_i} \quad (5)$$

K and ϵ are obtained from a solution of the modelled transport equations (see Launder and Spalding (1)):

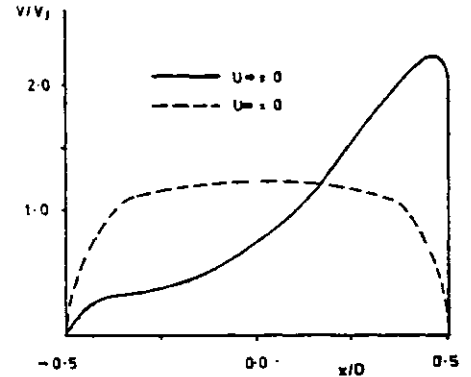


Fig. 2 V-Velocity profiles within the hole at $z/D = 0.0$ (From (2))

$$\bar{\rho} \bar{U}_i \frac{\partial K}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\frac{\mu_t}{\sigma_K} \frac{\partial K}{\partial x_i} \right] - \bar{\rho} \bar{u}_i \bar{u}_i \frac{\partial \bar{U}_i}{\partial x_i} - \rho \epsilon \quad (6)$$

$$\bar{\rho} \bar{U}_i \frac{\partial \epsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\frac{\mu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_i} \right] - c_\epsilon \frac{\epsilon}{K} \bar{\rho} \bar{u}_i \bar{u}_i \frac{\partial \bar{U}_i}{\partial x_i} - c_\epsilon \frac{\bar{\rho} \epsilon^2}{K} \quad (7)$$

The six empirical constants take the values:

$$C_\mu = 0.09 \quad C_1 = 1.44 \quad C_2 = 1.92 \quad \sigma_K = 1.0 \\ \sigma_\epsilon = 1.3 \quad \text{and} \quad \sigma_T = 0.9.$$

These have been chosen following extensive numerical experiment to give good agreement over a wide range of flows (Launder and Spalding (1)).

A procedure for solving the above equations was written by Dr W P Jones whilst in the employ of Rolls Royce Ltd. Only the basic features of this method can be described here. The procedure solves the equations by a linearised, implicit, conservative finite difference scheme, using a hybrid differencing technique. Central differences are used in regions where convective and diffusive fluxes are of similar order, whilst upwind differences are used in regions where convective fluxes dominate. The method is formulated in primitive variables using a staggered grid arrangement in which scalar quantities and pressure occupy the same grid locations, velocities being located mid-way between the pressures that drive them. A similar grid arrangement has been used by Amsden and Harlow (8). The procedure allows calculation on Cartesian or cylindrical polar grids of non-uniform spacing. It is therefore possible to concentrate the grid in the regions of the flow which require the fine grid. A difficulty with primitive variable formulation arises from the calculation of the pressure. It is necessary to obtain a pressure field such that the solution of the momentum equations satisfies continuity. This difficulty is overcome by the use of an equation for 'pressure correction' derived from a combination of the finite difference equations for momentum and continuity in a guess and correct procedure similar to that of Amsden and Harlow (8), Chorin (9) and Patankar and Spalding (10). The pressure and velocity fields are initialised and

Boundary	U	V	W	T	K	E
Wall	Couette flow relations	$V = 0$	Couette flow relations	$\partial T / \partial y = 0$ or h.t.c. from Couette flow	Couette flow relations	$L \propto y$ ($C = K^{1/2} / L$)
Hole	Hole calculation	Constant total pressure	Hole calculation	$T = T_j$	Hole calculation	Hole calculation
Top	$\partial u / \partial y = 0$	$V = 0$	$\partial w / \partial y = 0$	$\partial T / \partial y = 0$	$\partial K / \partial y = 0$	$\partial E / \partial y = 0$
Lateral (hole centre)	$\partial u / \partial z = 0$	$\partial v / \partial z = 0$	$W = 0$	$\partial T / \partial z = 0$	$\partial K / \partial z = 0$	$\partial E / \partial z = 0$
Lateral far body	$\partial u / \partial z = 0$	$\partial v / \partial z = 0$	$\partial w / \partial z = 0$	$\partial T / \partial z = 0$	$\partial K / \partial z = 0$	$\partial E / \partial z = 0$
Upstream	$U = U(y)$	$V = V(y)$	$W = 0$	$T = T_\infty$	$K = K(y)$	$E = E(y)$
Downstream	$\partial u / \partial x = 0$ plus factoring to maintain overall continuity	$\partial v / \partial x = 0$	$\partial w / \partial x = 0$	$\partial T / \partial x = 0$	$\partial K / \partial x = 0$	$\partial E / \partial x = 0$

Table. 1 Boundary conditions employed for the calculation of the jet-mainstream interaction

the three momentum equations solved to give a preliminary velocity field, which, in general, does not satisfy the continuity equation. The solution of the pressure correction equation provides the corrections to the velocity and pressure fields which drive the solution towards the mutual satisfaction of the momentum and continuity equations. This process is repeated on each iteration of the solution until a final converged solution is obtained.

COMPUTATIONAL DETAILS

As a result of the strong pressure gradients set up across the exit of the injection hole the flow within the hole becomes distorted. This effect was observed by Bergeles (2) who measured velocity profiles within a normal injection hole with cross-flow (Fig. 2). In order to account for the influence of this on the jet-mainstream interaction, the present calculations embody two three-dimensional elliptic flow solutions, one of the jet mainstream interaction and the other of the flow within the injection hole. The two calculations are iterated together to provide converged solutions of the flow.

Calculation of the Jet-Mainstream Interaction

The jet-mainstream interaction is modelled on a rectangular grid. Boundary conditions are required on all six boundaries of the domain and these are summarised in Table 1.

The definition of the circular injection hole by the rectangular grid poses problems. In the present study where the hole boundary crosses the face of one of the calculation cells, the surface area of that face is modified so that the correct fluxes of mass and momentum from the hole are conserved.

The linkage between the calculation within the hole and that of the external flow is provided by making use of the observation of Bergeles (2) that the velocity profile across the injection hole is

well approximated by that given by assuming constant total pressure across the hole exit. This assumption is used to provide the V-velocity distribution across the exit of the injection hole in response to the pressure field arising from the external flow calculation. These distributions are used as exit boundary conditions for the hole calculations, which in turn give the distributions of U, W, K, and E across the hole exit. Details of the linking between the hole and jet-mainstream calculation are given in the second part of this section.

In order to make the best use of the available finite difference grid, it is necessary to reduce the calculation domain to the minimum possible size. This is achieved by completing a series of calculations in which each boundary location is altered in turn by adding or removing grid lines. The minimum calculation domain size is defined by the boundary locations beyond which the solution ceases to change with additional grid lines. The size of the domain required is dependent on the jet to free-stream momentum ratio, J, so that this exercise was completed at two momentum ratios, 0.058 and 4.0, equivalent to the highest and lowest conditions studied, with the results shown in Table. 2.

Boundary		$J = 0.0576$	$J = 4.0$
Upstream	X_j/D	-2.0	-2.0
Downstream	X_p/D	5.5	5.5
Lateral	Z/D	2.0	3.0
Top	Y/D	4.0	6.0

Table. 2 Boundary locations determined for the calculation of the jet-mainstream interaction

Boundary	U_0	V_0	W_0	K	$\mathcal{E}$
Wall	$U_0 = 0$	Couette Flow Relations			$L \propto y$
Hole Axis	Mean of Adjacent Values				
Symmetry Plane through Hole Centre	$\partial U_0 / \partial y = 0$	$\partial V_0 / \partial y = 0$	$W_0 = 0$	$\partial K / \partial y = 0$	$\partial \mathcal{E} / \partial y = 0$
Upstream	$U_0 = 0$	$V_0 = V_0(r)$	$W_0 = 0$	$K = K(r)$	$\mathcal{E} = \mathcal{E}(r)$
Downstream	Linear Extrapolation	From External Solution	Linear Extrapolation		

Table. 3 Boundary conditions employed for the calculation within the hole

The grid dependence of the solutions was studied by completing a series of six calculations in which refinement was completed in each of the X, Y and Z directions in turn. The finest grid employed, of $25 \times 20 \times 15$ nodes ($X \times Y \times Z$), still showed evidence of grid dependence, however this represented the practical limit of refinement on the available computer facilities. The effect of grid refinement on the U-velocity profile downstream of the hole is shown in Fig. 3. It can be seen that as the grid is refined the velocity profile starts to develop a peak. The V-velocity profiles downstream of the hole also show changes in the later stages of refinement.

It is important to study the effect of refinement of each grid direction in turn, since the refinement of individual components of the grid can give rise to opposing effects. It would therefore be possible to obtain spurious grid independence where, as is the normal practice, grid refinement is carried out in all directions simultaneously.

Calculation within the Hole

The calculations within the injection hole employ a cylindrical co-ordinate system with boundary conditions as summarised in Table. 3.

The distribution of the V_0 -velocity component and pressure across the hole exit are derived from the calculation of the jet-mainstream interaction, whereas the distributions of U_0 , W_0 , K and $\mathcal{E}$ are obtained by linear extrapolation from the calculation planes immediately upstream of the hole exit.

The extent of the elliptic region within the hole is dependent on the distortion at the hole exit. Since the maximum distortion occurs at the lowest blowing rates, calculations to determine the upstream boundary location and grid dependence were set up using Bergeles (2) measured exit velocity profile for a blowing rate of 0.1. From these calculations it was determined that the elliptic region extended 1.5 hole diameters into the hole. Calculations on grids of $8 \times 9 \times 12$ nodes were found to give accurate solutions, with a CPU requirement of only 30 seconds on an IBM 370-168.

The iteration between the calculation of the jet-mainstream interaction and that within the hole follows the following sequence:

1 Several iterations of the jet-mainstream interaction calculation are completed, with the U and V velocity components across the hole exit set to zero. The V-velocity profile across the hole is

allowed to develop through the constant total pressure boundary condition.

2 Once this solution starts to become established the calculation is stopped. The calculation of the flow in the hole is then started and iterated until a converged solution is obtained.

3 The calculation of the jet-mainstream interaction is then restarted, using the distributions of the U and V-velocity components and K and $\mathcal{E}$ at the hole exit arising from the solution within the hole.

Three cycles of hole-external flow calculations are generally sufficient to obtain a converged solution, although at low blowing rates more cycles may be required. A full solution with three cycles requires 800K bytes storage and 50 minutes CPU on an IBM 370-168.

RESULTS AND DISCUSSION

The importance of calculating the distortion of the flow within the injection hole may be determined from Fig.4 in which comparisons between the measured velocity profile data of Bergeles (2) and the results of two predictions are presented. In one of the

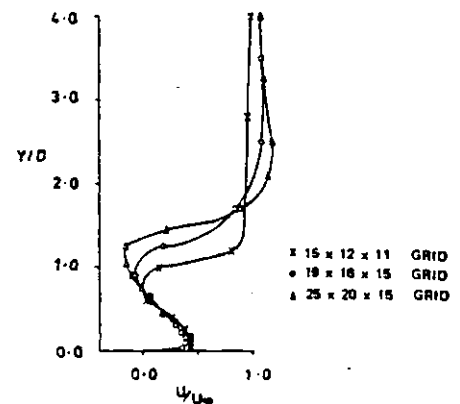


Fig. 3 Effect of grid refinement on the U-velocity profile at $X/D = 1.4$, $Z/D = 0.0$

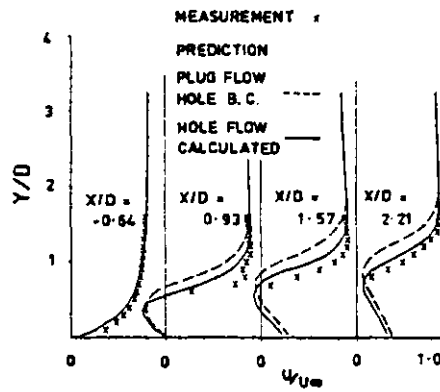


Fig. 4 Comparison of predicted U-velocity profiles for $M = 1.0$ with measurements of (2)

predictions a plug flow boundary condition across the hole exit is employed, whilst the present procedure with calculation within the hole is employed in the other prediction. The calculation of the development of the flow within the hole has a significant effect on the solution, resulting in a marked improvement in the agreement with the measured data.

The present arrangements for linking the calculation within the hole and that of the jet-mainstream interaction are based on the simple assumption of constant total pressure across the hole exit. A more rigorous treatment of this linking could be devised, however the present method does accurately represent the jet velocity profile. The recent Laser Doppler anemometry measurements obtained by Crabb (5) provide convincing evidence for this. A comparison between the jet exit velocity profiles measured by Crabb and those obtained by the present predictions is shown in Fig. 5, from which it can be

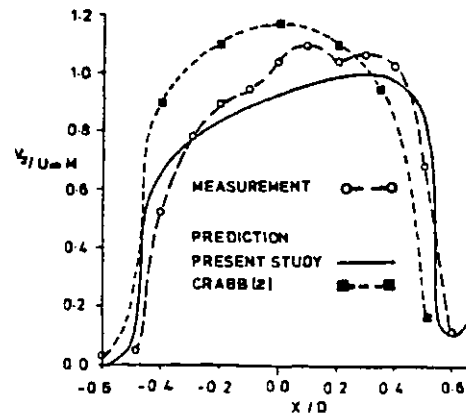


Fig. 5 Comparison of predicted jet exit velocity profiles for $M = 2.5$ with measurements of (5)

6

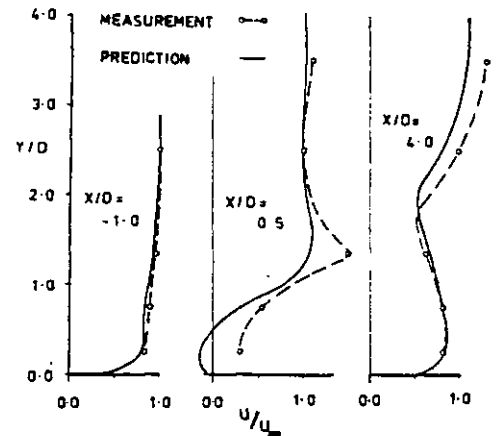


Fig. 6 Comparison of predicted U-velocity profiles for $M = 2.5$ with measurements of (5)

seen that excellent agreement has been obtained. Included in this figure is the prediction obtained by Crabb employing a similar basic numerical method to the present work, but with a hole exit boundary condition based on fully developed turbulent pipeflow profiles. The level of agreement between measurement and prediction is significantly better with the present predictions than with those of Crabb.

Further comparisons with the measurements and predictions of Crabb are presented in Figs. 6 and 7. In general, the present predictions are in good agreement with the measured U-velocity profiles, which are shown in Fig. 6, however the predictions show a region of reverse flow immediately downstream of the hole which is not shown in the measurements, and under predict the peak velocity downstream of the hole. Good agreement is also obtained with the V-velocity

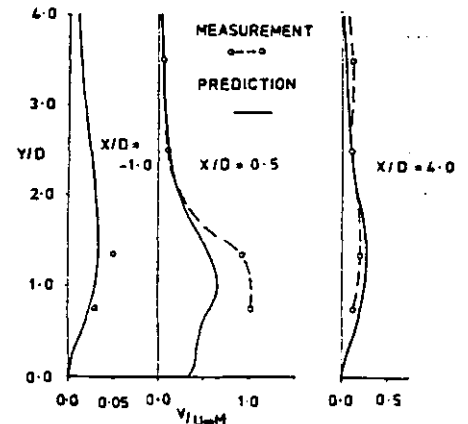


Fig. 7 Comparison of predicted V-velocity profiles for $M = 2.5$ with measurements of (5)

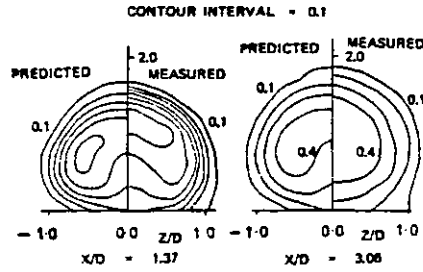


Fig. 8 Comparison of predicted effectiveness contours for $M = 1.0$ with measurements of (7)

profiles shown in Fig. 7, the predictions showing a tendency to overpredict the velocity peak near the wall, and underpredict the velocity peak far from the wall.

In the immediate vicinity of the hole the observed discrepancies may be due to the manner in which the hole is represented by the rectangular grid. Further downstream the discrepancies in the U-velocity profile are largely the result of inadequate grid refinement: as is shown in Fig. 3, the peak in the U-velocity profile starts to develop as the grid is refined. Part of the discrepancy may result from the assumptions implicit in the $K-\epsilon$ turbulence model, however with the present calculations it is not possible to assess the accuracy of the turbulence modelling, since the solutions are still grid dependent. Such an assessment can only be carried out on grid independent solutions.

Comparisons with the non-dimensional temperature (effectiveness) contour plots of Ramsey and Goldstein (7) are presented in Fig. 8. Apart from the region in the vicinity of the wall, the jet spread and penetration are well predicted. A feature of the predictions which is clearly displayed in these plots is the overprediction of the streamwise vortices downstream of

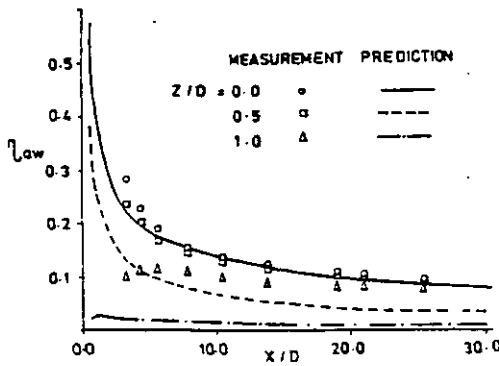


Fig. 9 Comparison of predicted adiabatic wall effectiveness for $M = 1.006$ with measurements of (11)

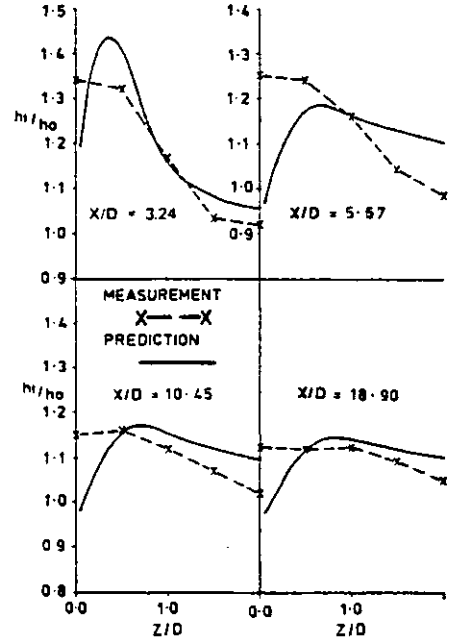


Fig. 10 Comparison of predicted heat transfer coefficients for $M = 1.013$ with measurements of (11)

the hole. This is consistent with the overprediction of the near-wall peak in the V-velocity profiles shown in Fig. 7. Unfortunately the variation in V-velocity with increasing grid refinement does not follow consistent trends so that it is not possible to determine whether this arises from the turbulence modelling or inadequate grid definition.

The overprediction of the streamwise vortices has important consequences for the conditions at the wall. This is clearly shown on the predictions of adiabatic wall effectiveness and wall heat transfer coefficients, comparisons of which, with the measurements of Eriksen (11), are shown in Figs. 9 and 10. The adiabatic wall effectiveness comparisons show fair agreement in the jet centreline, however the lateral spread of the effectiveness is significantly underpredicted, reflecting the additional mainstream fluid entrained under the jet as a result of the overprediction of the streamwise vortices.

The overall level of agreement between the measured and predicted heat transfer coefficients shown in Figure 10 is good. However, the centre-line heat transfer coefficient is consistently underpredicted, whilst there is a tendency to overpredict the heat transfer for lateral locations more than about $0.7D$ off the centre-line. This too can be interpreted in terms of the overprediction of the streamwise vortices. At off centre-line locations, the vortex causes the flow to take on the character of a reattachment region, with which we associate high heat transfer rates; along the jet centre-line, the flow separates giving rise to suppressed heat transfer rates. Clearly

overprediction of the streamwise vortices will exaggerate these effects, however this is insufficient to account for the full discrepancy on the jet centre-line. The additional underprediction of the heat transfer rate arises from the use of wall functions based on boundary layer flows to calculate the heat transfer at the wall (see Launder and Spalding (1)). Such wall functions assume Couette flow near the wall which will be invalid on the centre-line where there are large velocity components away from the wall.

CONCLUSIONS

1 Accurate predictions of the jet in cross-flow can only be achieved if the development of the flow within the hole is calculated. The method described in this paper provides a means of achieving this.

2 The simple constant total pressure assumption employed to link the calculation within the hole with that of the jet-mainstream interaction accurately represents the jet exit velocity profiles.

3 Many features of the flow are well represented by the present predictions. For example the jet spread and penetration show excellent agreement with the measurements of Ramsey and Goldstein (7), and the U and V-velocity profiles show good agreement with measured data. However the solutions still exhibit signs of grid dependence this being the major cause of the underprediction of the peak in the U-velocity profile downstream of the hole.

4 The strength of the streamwise vortices downstream of the hole is overpredicted. This affects the predictions of the adiabatic wall effectiveness and the wall heat transfer rate. The cause of the overprediction is not clear; it may be another result of inadequate grid refinement, or arise from the assumptions involved in the $k-\epsilon$ turbulence model.

5 The use of a heat flux wall function based on boundary layer type flows with Couette-like flow near the wall is invalid near the jet centre-line where there are large velocity components away from the wall. This contributes to the underprediction of the heat transfer coefficients along the jet centre-line.

6 It is important to carry out grid refinement in each grid direction in turn, since the refinement of the different grid directions can give rise to opposing effects. Where grid refinement is carried out in all directions simultaneously it would be possible to, at best, gain a misleading impression of the grid dependence of the solutions and, at worst, obtain a spurious grid independence.

RECOMMENDATIONS FOR FURTHER WORK

The present predictions show very encouraging agreement with measured data, however further improvement is clearly still possible. The accuracy of the solutions should be improved with further grid refinement. The predictions presented were completed some time ago on an IBM 370-168. Finer grid calculations are possible on present computer systems and this situation should improve with the growth of computing power in the future.

The present linking between the hole and jet-mainstream interaction restricts the method to normal holes with large length to diameter ratios. A more rigorous treatment of this should allow predictions on inclined holes with small length-diameter ratios.

Whilst the present heat transfer predictions

are encouraging, improvement should result from matching in the laminar sublayer, hence removing the need for wall functions with their attendant limitations. Such calculations require fine calculation grids near the wall and are not yet practicable on present computer systems.

ACKNOWLEDGEMENTS

In completing this work I am indebted to Professor J H Whitelaw and Dr A D Gosman of Imperial College, London for their support and guidance throughout my studies of which this work forms a part. I also wish to express my thanks to the Ministry of Defence and Rolls Royce Ltd for allowing me to publish this work. Finally, I would like to acknowledge Mr H Tubbs and Mr B Barry of Rolls Royce Ltd whose support made this project possible.

REFERENCES

- 1 Launder, B.E. and Spalding, D.B., "The Numerical Computations of Turbulent Flows," Imperial College, Mech Eng Report, HTS/73/2, 1973.
- 2 Bergeles, G., "Three-Dimensional Discrete-Hole Cooling Processes, an Experimental and Theoretical Study," Ph.D Thesis, University of London, July 1976.
- 3 Chien, J.C. and Schetz, J.A., "Numerical Solution of the Three-Dimensional Navier-Stokes Equations with Applications to Channel Flows and a Buoyant Jet in a Cross-Flow," Trans. ASME Jnl of App. Mech Vol 42, September 1975, pp. 575-579.
- 4 Patankar, S.V., Basu, D.K., Alpay, S.A., "Prediction of the Three-Dimensional Velocity Field of a Deflected Turbulent Jet," Trans. ASME Jnl of Fluids Engineering, Vol 99, Dec. 1977, pp. 758-762.
- 5 Crabb, D., "Jets in Crossflow," Ph.D Thesis, University of London, 1979.
- 6 Jones, W.P. and McGuirk, J.J., "Computation of a Round Turbulent Jet Discharging into a Confined Crossflow," presented at the Second Symposium on Turbulent Shear Flows, Imperial College, July 1979.
- 7 Ramsey, J.W. and Goldstein, R.J., "Interaction of a Heated Jet with a Deflecting Stream," Trans. ASME Jnl of Heat Transfer, Vol 93, Nov 1971, pp. 365-372.
- 8 Amaden, A.A. and Harlow, F.H., "The SMAC Method," Los Alamos Scientific Lab. Report LA-4370, 1970.
- 9 Chorin, A.J., "Numerical Solution of the Navier-Stokes Equations," Math. Comp., Vol 22, 1968, p. 745.
- 10 Patankar, S.V. and Spalding, D.B., "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows," Int. Jnl of Heat and Mass Transfer, Vol 15, No.10, Oct 1972, pp. 1787-1805.
- 11 Eriksen, V.L., "Film Cooling Effectiveness and Heat Transfer with Injection through Holes," NASA CR-72991, August 1971.

Figure Captions for Chapter 1

- Fig. 1.1 Typical turbine rotor blade profile and
Mach number distribution
- Fig. 1.2 Flow from a two-dimensional flush slot
- Fig. 1.3 Flowfield in the vicinity of a jet in crossflow

Figure Captions for Chapter 2

- Fig. 2.1(a) Experimental arrangement
- Fig. 2.1(b) Ejection configurations
- Fig. 2.2 Static tapping arrangement on surface
 of cylinder
- Fig. 2.3 Secondary air supply system
- Fig. 2.4 (a) Pitot/Tracer gas sampling probe
- Fig. 2.4(b) Five-hole probe
- Fig. 2.5 Definition sketch for five-hole probe
 measurements

Figure Captions for Chapter 3

- Fig. 3.1 Static pressure distribution on cylinder
without ejection
- Fig. 3.2 Velocity profiles for cylinder without ejection
- Fig. 3.3 Flow visualisations for $\Theta_s = 0^\circ$
- Fig. 3.4 Static pressure distribution, $\Theta_s = 0^\circ$, $J = 1.98$
- Fig. 3.5(a) Velocity profiles, $\Theta_s = 0^\circ$, $J = 0.89$, $Z/D = 0.0$
- Fig. 3.5(b) Velocity profiles, $\Theta_s = 0^\circ$, $J = 8.89$, $Z/D = 1.0$
- Fig. 3.6 Concentration profiles, $\Theta_s = 0^\circ$, $J = 0.89$
- Fig. 3.7 Surface concentration, $\Theta_s = 0^\circ$, $J = 0.99$
- Fig. 3.8 Concentration profiles, $\Theta_s = 0^\circ$, $J = 0.48$
- Fig. 3.9 Surface concentration, $\Theta_s = 0^\circ$, $J = 0.48$
- Fig. 3.10 Flow visualisations for $\Theta_s = \pm 20^\circ$
- Fig. 3.11(a) Velocity profiles, $\Theta_s = \pm 20^\circ$, $J = 1.98$, $Z/D = 0.0$
- Fig. 3.11(b) Velocity profiles, $\Theta_s = \pm 20^\circ$, $J = 1.98$, $Z/D = 1.0$
- Fig. 3.12 Concentration profiles, $\Theta_s = \pm 20^\circ$, $J = 1.98$
- Fig. 3.13 Surface concentration, $\Theta_s = \pm 20^\circ$, $J = 1.98$
- Fig. 3.14 Concentration profiles, $\Theta_s = \pm 20^\circ$, $J = 1.11$
- Fig. 3.15 Surface concentration, $\Theta_s = \pm 20^\circ$, $J = 1.11$
- Fig. 3.16 Concentration profiles, $\Theta_s = \pm 20^\circ$, $J = 0.55$
- Fig. 3.17 Surface concentration, $\Theta_s = \pm 20^\circ$, $J = 0.55$
- Fig. 3.18 Flow visualisations for $\Theta_s = \pm 40^\circ$
- Fig. 3.19(a) Velocity profiles, $\Theta_s = \pm 40^\circ$, $J = 1.81$, $Z/D = 0.0$
- Fig. 3.19(b) Velocity profiles, $\Theta_s = \pm 40^\circ$, $J = 1.81$, $Z/D = 1.0$
- Fig. 3.20 Concentration profiles, $\Theta_s = \pm 40^\circ$, $J = 1.81$
- Fig. 3.21 Surface concentration, $\Theta_s = \pm 40^\circ$, $J = 1.81$
- Fig. 3.22 Concentration profiles, $\Theta_s = \pm 40^\circ$, $J = 1.0$
- Fig. 3.23 Surface concentration, $\Theta_s = \pm 40^\circ$, $J = 1.0$
- Fig. 3.24 Concentration profiles, $\Theta_s = \pm 40^\circ$, $J = 0.5$
- Fig. 3.25 Surface concentration, $\Theta_s = \pm 40^\circ$, $J = 0.5$

Figure Captions for Chapter 4

- Fig. 4.1(a) Three-dimensional control volume
Fig. 4.1(b) Specification of ϕ and V_z cells
Fig. 4.2 Cell with solid surface boundary

Figure Captions for Chapter 5

- Fig. 5.1 Calculation domain for flush slot computations
- Fig. 5.2 Grid form employed for flush slot computations
- Fig. 5.3 Velocity profile in exit region of secondary jet, $J = 0.24$, $Z/D = 0.0$
- Fig. 5.4 Effect of grid refinement, $\alpha = 60^\circ$, $J = 1.45$
- Fig. 5.5 Reattachment distance downstream of slot
- Fig. 5.6(a) Profiles of terms in the axial momentum equation, $\alpha = 90^\circ$, $J = 1.24$
- Fig. 5.6(b) Profiles of terms in the equation for the conserved scalar, T , $\alpha = 90^\circ$, $J = 1.24$
- Fig. 5.7 Estimation of the regions of influence of streamline curvature on turbulent mixing
- Fig. 5.8 Velocity profiles, $\alpha = 90^\circ$
- Fig. 5.9 Adiabatic wall effectiveness, $\alpha = 90^\circ$
- Fig. 5.10 Stanton number, $\alpha = 90^\circ$
- Fig. 5.11 Velocity profiles, $\alpha = 60^\circ$
- Fig. 5.12 Adiabatic wall effectiveness, $\alpha = 60^\circ$
- Fig. 5.13 Velocity profiles, $\alpha = 30^\circ$
- Fig. 5.14 Impervious wall effectiveness, $\alpha = 30^\circ$

Figure Captions for Chapter 6

- Fig. 6.1 Calculation domain for the jet-mainstream interaction
- Fig. 6.2 Effect of grid refinement on the V_x - velocity profile, $J = 2.0$, $x/D = 1.4$, $Z/D = 0.0$
- Fig. 6.3(a) Profiles of terms in the axial momentum equation, $J = 1.0$, $Z/D = 0.25$
- Fig. 6.3(b) Profiles of terms in the equation for the conserved scalar, C , $J = 1.0$, $Z/D = 0.25$
- Fig. 6.4 Influence of hole exit boundary condition on V_x - velocity profiles
- Fig. 6.5 Domain and co-ordinate system for computation within the hole
- Fig. 6.6 Velocity profiles in hole exit plane, $J = 1.0$, $Z/D = 0.0$
- Fig. 6.7 V_y - velocity profile through jet, $J = 2.3$, $y/D = 0.25$, $Z/D = 0.0$
- Fig. 6.8 Velocity profiles, $J = 2.3$, $Z/D = 0.0$
- Fig. 6.9(a) Velocity profiles, $J = 2.0$, $Z/D = 0.0$
- Fig. 6.9(b) Velocity profiles, $J = 1.0$, $Z/D = 0.0$
- Fig. 6.10(a) Velocity profiles, $J = 1.0$, $Z/D = 0.0$
- Fig. 6.10(b) Velocity profiles, $J = 0.5$, $Z/D = 0.0$
- Fig. 6.10(c) Velocity profiles, $J = 0.24$, $Z/D = 0.0$
- Fig. 6.11 Effectiveness contours on cross-stream planes
- Fig. 6.12 Adiabatic wall effectiveness
- Fig. 6.13 Surface heat transfer coefficients, $J = 1.990$, $p_5/p_\infty = 1.0$
- Fig. 6.14 Surface heat transfer coefficients, $J = 1.013$, $p_5/p_\infty = 1.0$
- Fig. 6.15 Surface heat transfer coefficients, $J = 0.544$, $p_5/p_\infty = 1.0$

Figure Captions for Chapter 7

- Fig. 7.1 Definition of plane of symmetry
- Fig. 7.2 Schematic diagram of embedded grid system
- Fig. 7.3(a) Rectangular grid : Preliminary V_x - cell interface
- Fig. 7.3(b) Rectangular grid : Preliminary V_y - cell interface
- Fig. 7.3(c) Rectangular grid : V_z and scalar cell interface
- Fig. 7.4 Domain for rectangular grid laminar flow solutions
- Fig. 7.5 Cylinder with afterbody
- Fig. 7.6 Cylindrical grid solution domain in the $r - \theta$ plane
- Fig. 7.7 V_θ - velocity profiles for $Re_D = 40$
- Fig. 7.8 Velocity profiles for cylinder without ejection
- Fig. 7.9 Static pressure distribution on cylinder without ejection
- Fig. 7.10 Solution domain for cylindrical grid computations with ejection
- Fig. 7.11 Effect of grid refinement on the cylindrical grid solution, $\theta_r = \pm 20^\circ$, $J = 2.0$, $Z/D = 0.0$
- Fig. 7.12(a) Profiles of terms in the radial momentum equation, $\theta_r = \pm 40^\circ$, $J = 1.81$, $Z/D = 0.25$
- Fig. 7.12(b) Profiles of terms in the tangential momentum equation, $\theta_r = \pm 40^\circ$, $J = 1.81$, $Z/D = 0.25$
- Fig. 7.12(c) Profiles of terms in the equation for the conserved scalar, C , $\theta_r = \pm 40^\circ$, $J = 1.81$, $Z/D = 0.25$
- Fig. 7.13 Impervious wall effectiveness, $\theta_r = 0^\circ$, $Z/D = 0.0$
- Fig. 7.14 Predicted flow regimes for $\theta_r = 0^\circ$
- Fig. 7.15(a) Velocity profiles, $\theta_r = 0^\circ$, $J = 0.89$, $Z/D = 0.0$
- Fig. 7.15(b) Velocity profiles, $\theta_r = 0^\circ$, $J = 0.89$, $Z/D = 1.0$
- Fig. 7.16(a) Velocity profiles, $\theta_r = \pm 20^\circ$, $J = 1.98$, $Z/D = 0.0$
- Fig. 7.16(b) Velocity profiles, $\theta_r = \pm 20^\circ$, $J = 1.98$, $Z/D = 1.0$

Figure Captions for Chapter 7 (cont.)

Fig. 7.17(a) Velocity profiles, $\theta_{\gamma} = \pm 40^{\circ}$, $J = 1.81$, $Z/D = 0.0$

Fig. 7.17(b) Velocity profiles, $\theta_{\gamma} = \pm 40^{\circ}$, $J = 1.81$, $Z/D = 1.0$

Fig. 7.18 Concentration profiles, $\theta_{\gamma} = 0^{\circ}$, $J = 0.89$

Fig. 7.19(a) Concentration profiles, $\theta_{\gamma} = \pm 20^{\circ}$, $J = 1.98$

Fig. 7.19(b) Surface concentration, $\theta_{\gamma} = \pm 20^{\circ}$, $J = 1.98$

Fig. 7.20(a) Concentration profiles, $\theta_{\gamma} = \pm 40^{\circ}$, $J = 1.81$

Fig. 7.20(b) Surface concentration, $\theta_{\gamma} = \pm 40^{\circ}$, $J = 1.81$

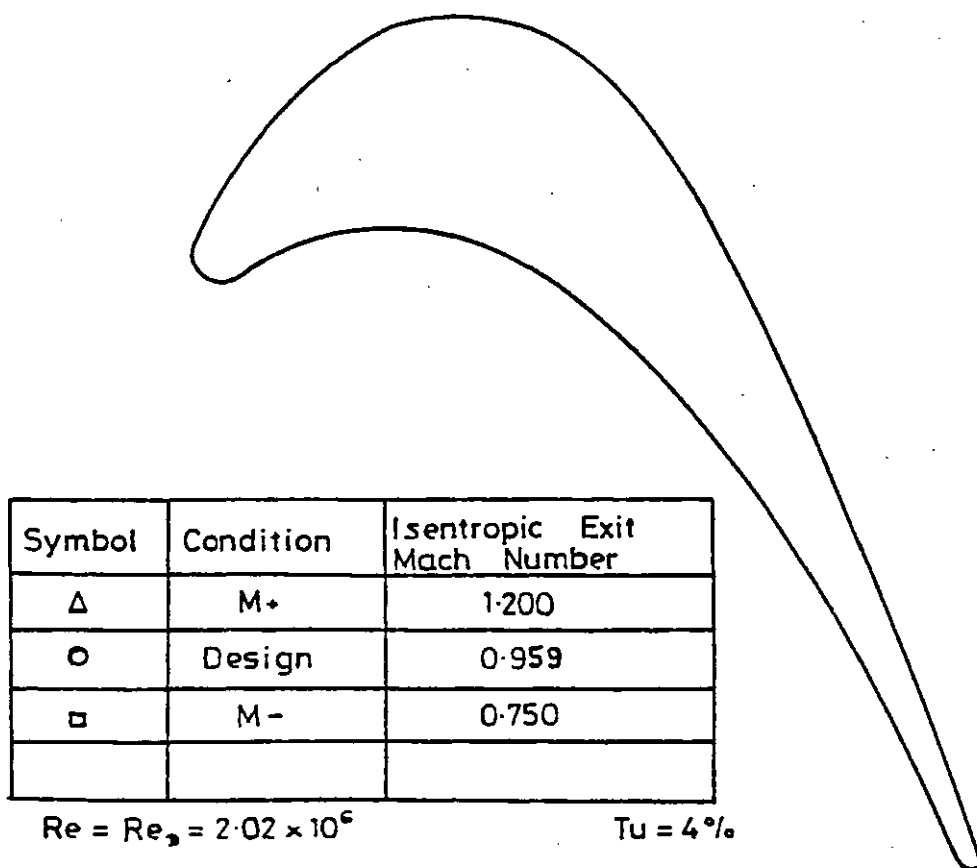
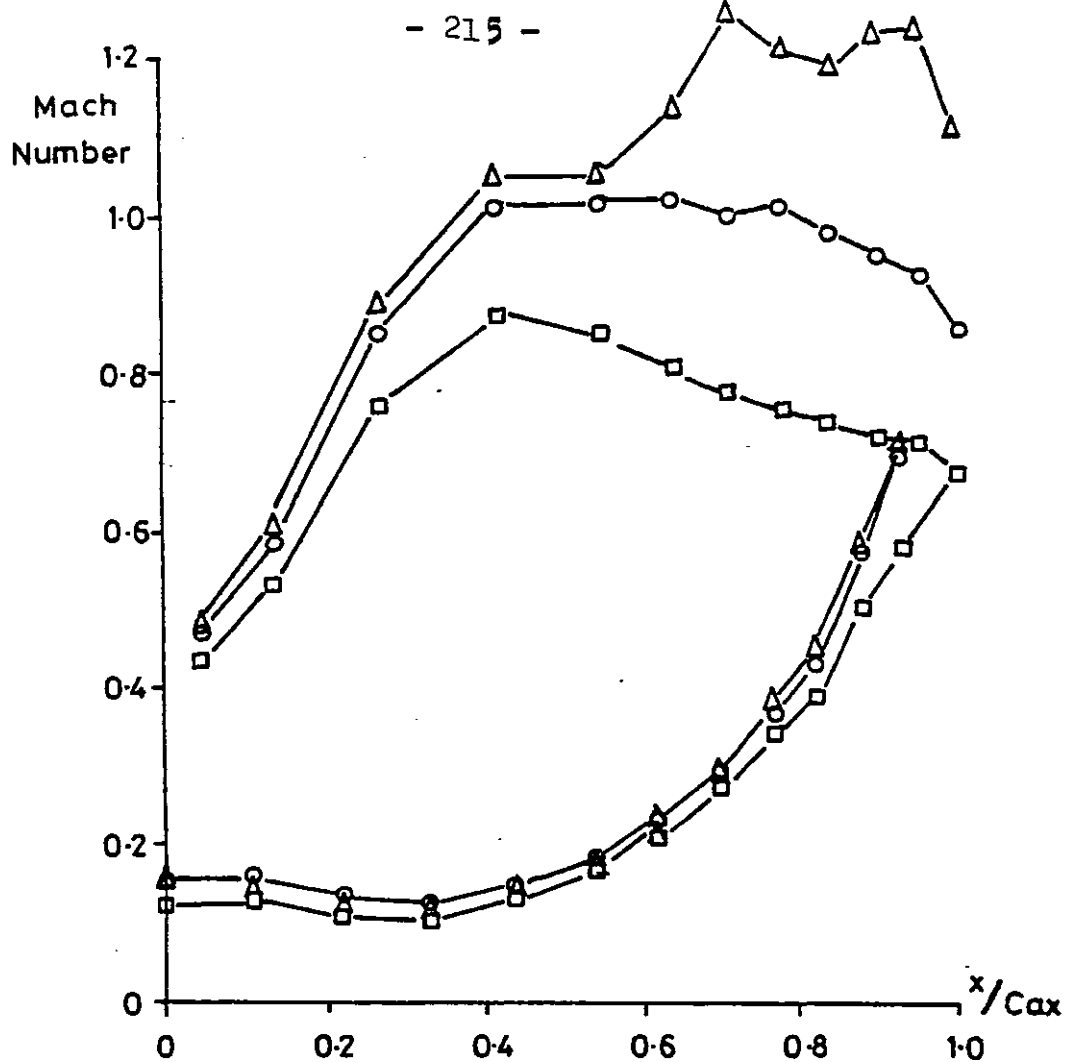


Fig. 1.1 Typical turbine rotor blade profile and Mach number distribution

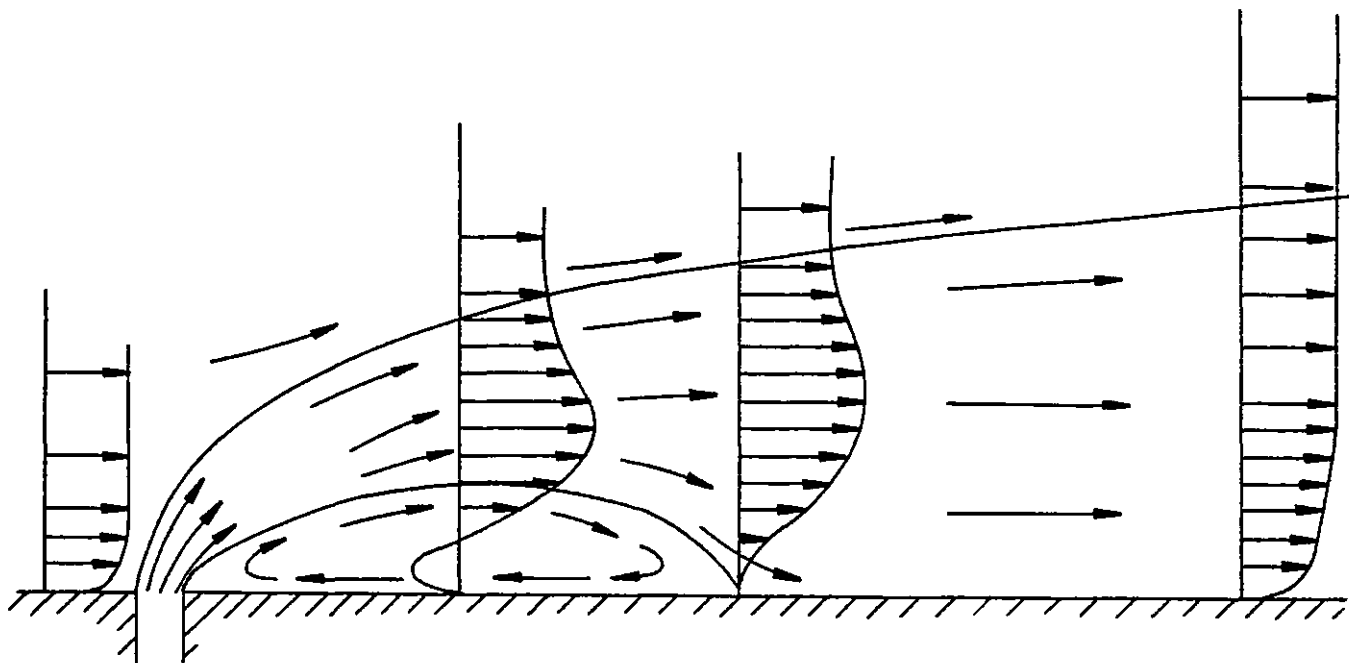


Fig. 1.2 Flow from a two-dimensional flush slot

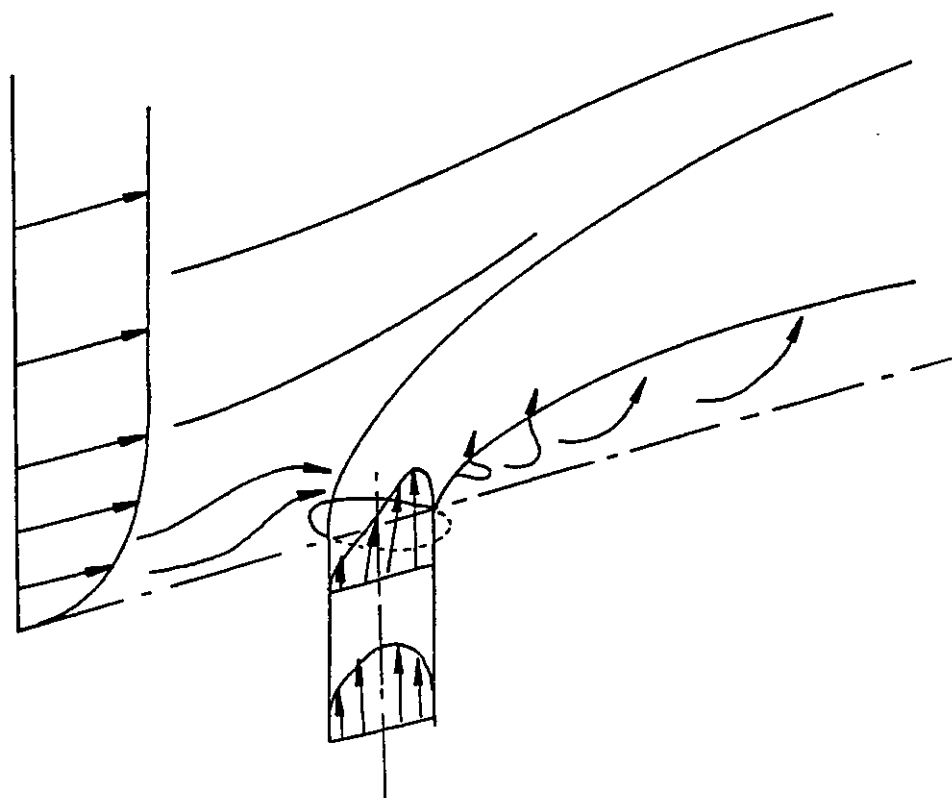


Fig. 1.3 Flowfield in the vicinity of a jet in crossflow

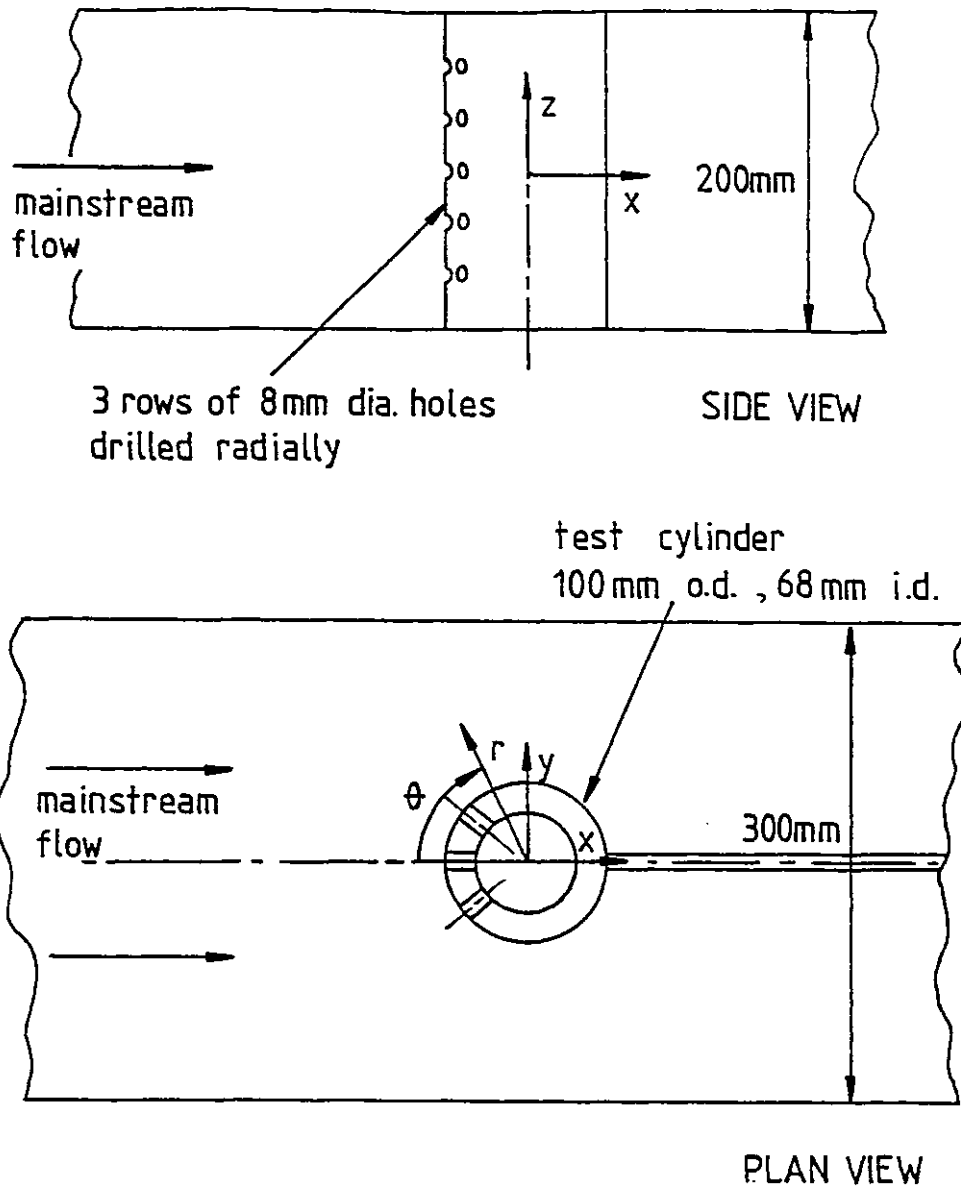


Fig. 2.1(a) Experimental arrangement

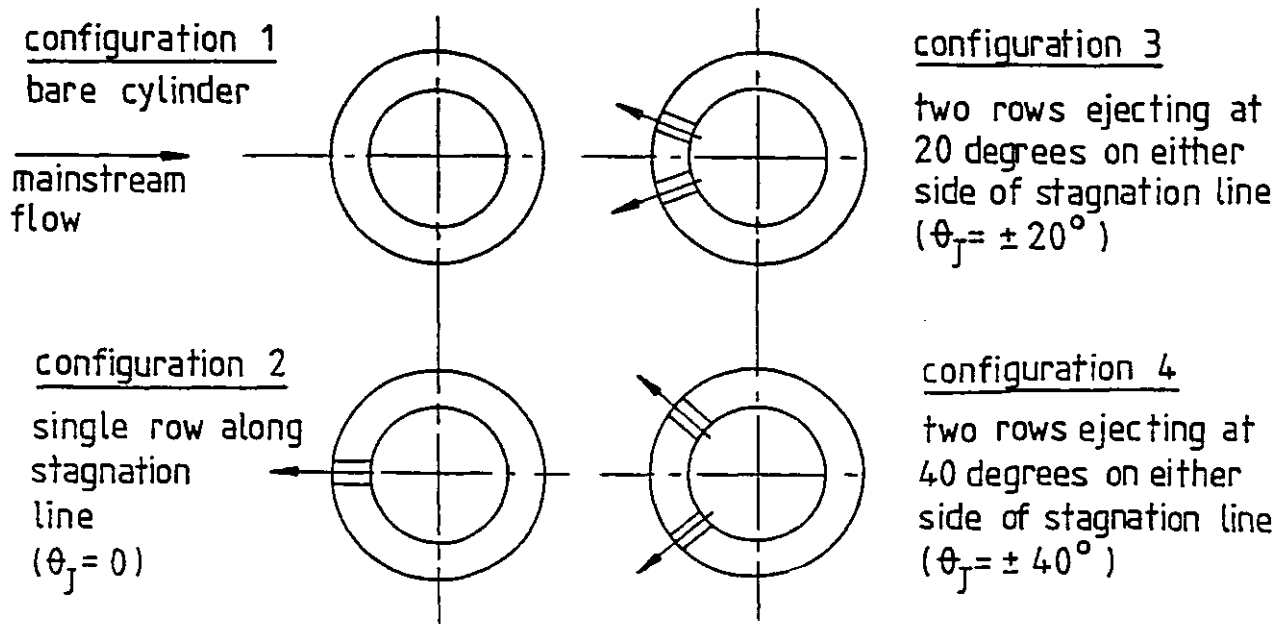


Fig. 2.1(b) Ejection configurations

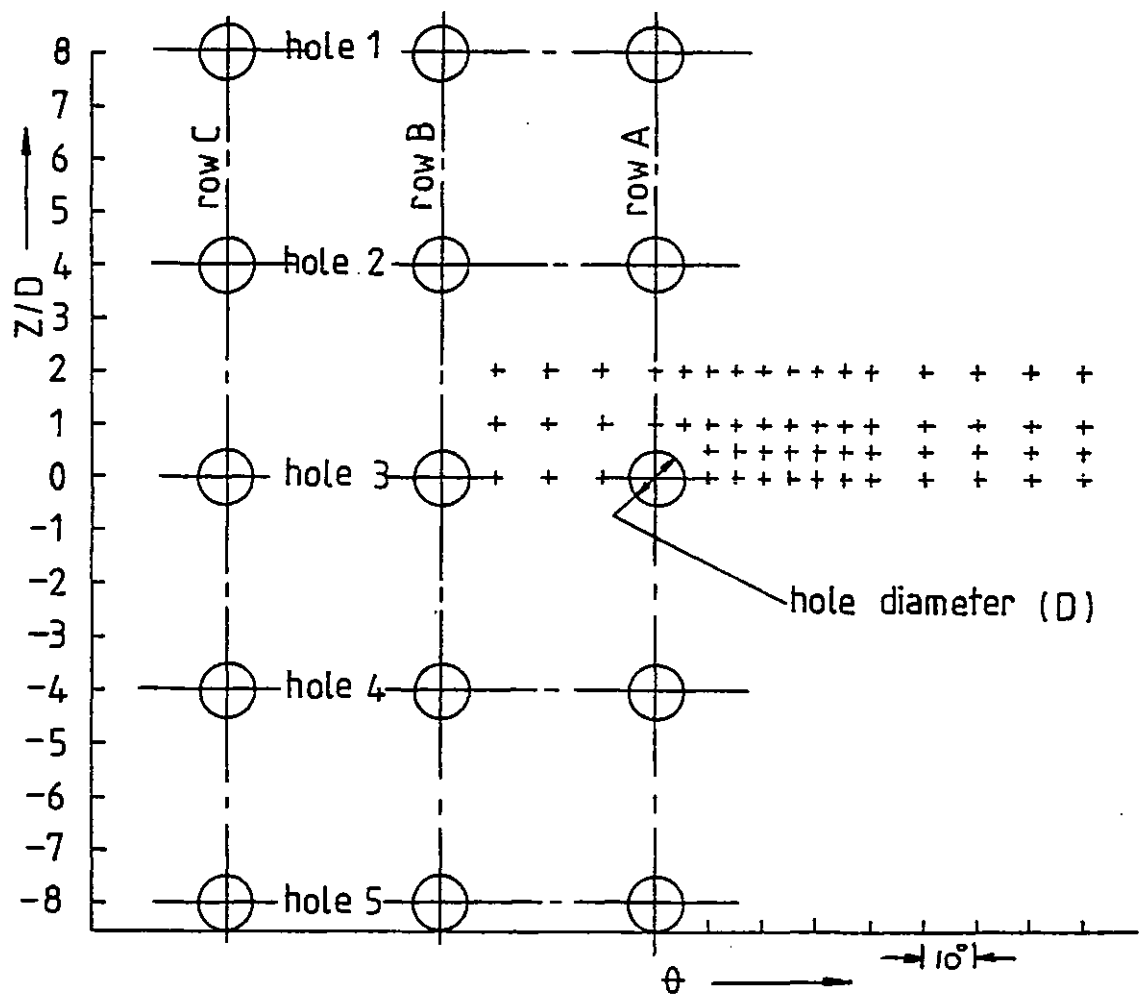


Fig. 2.2 Static tapping arrangement on surface of cylinder

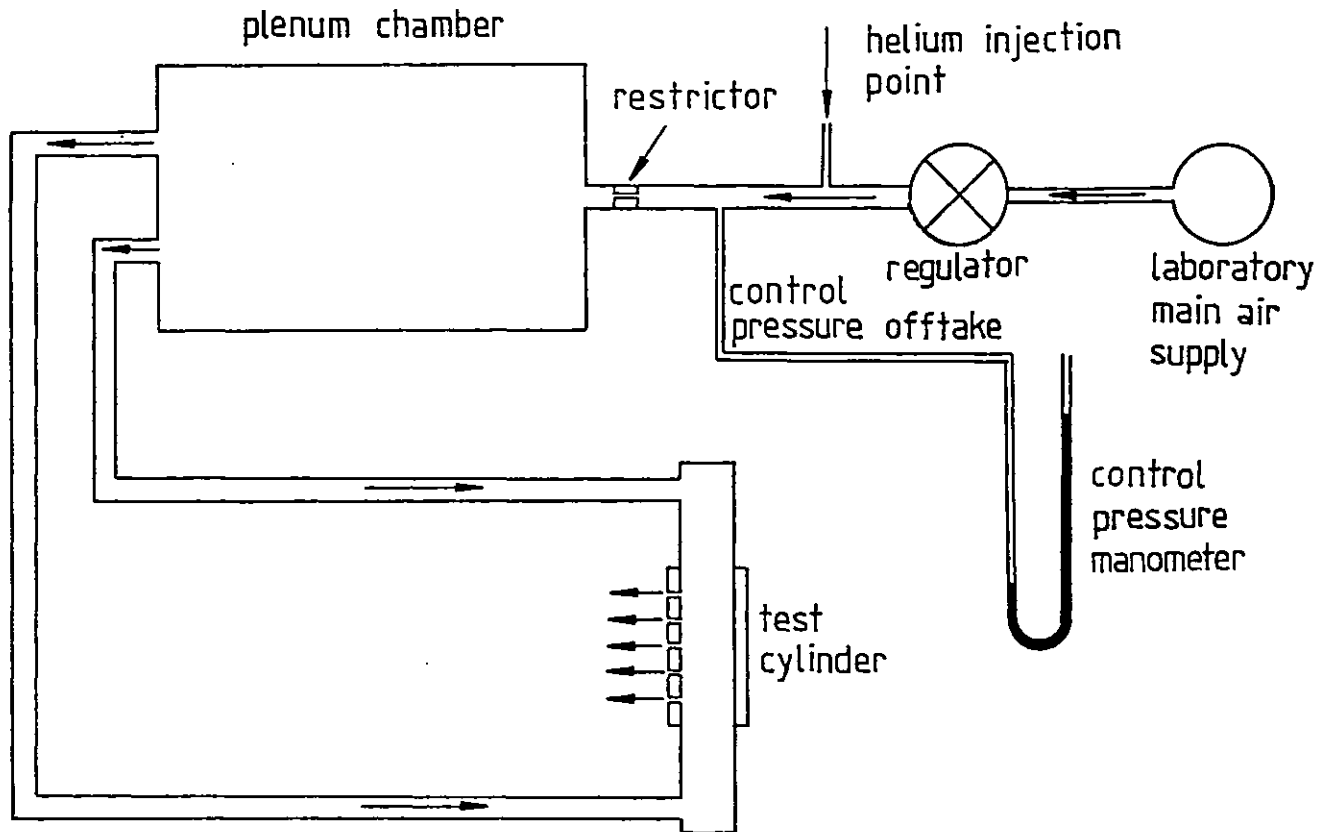


Fig. 2.3 Secondary air supply system

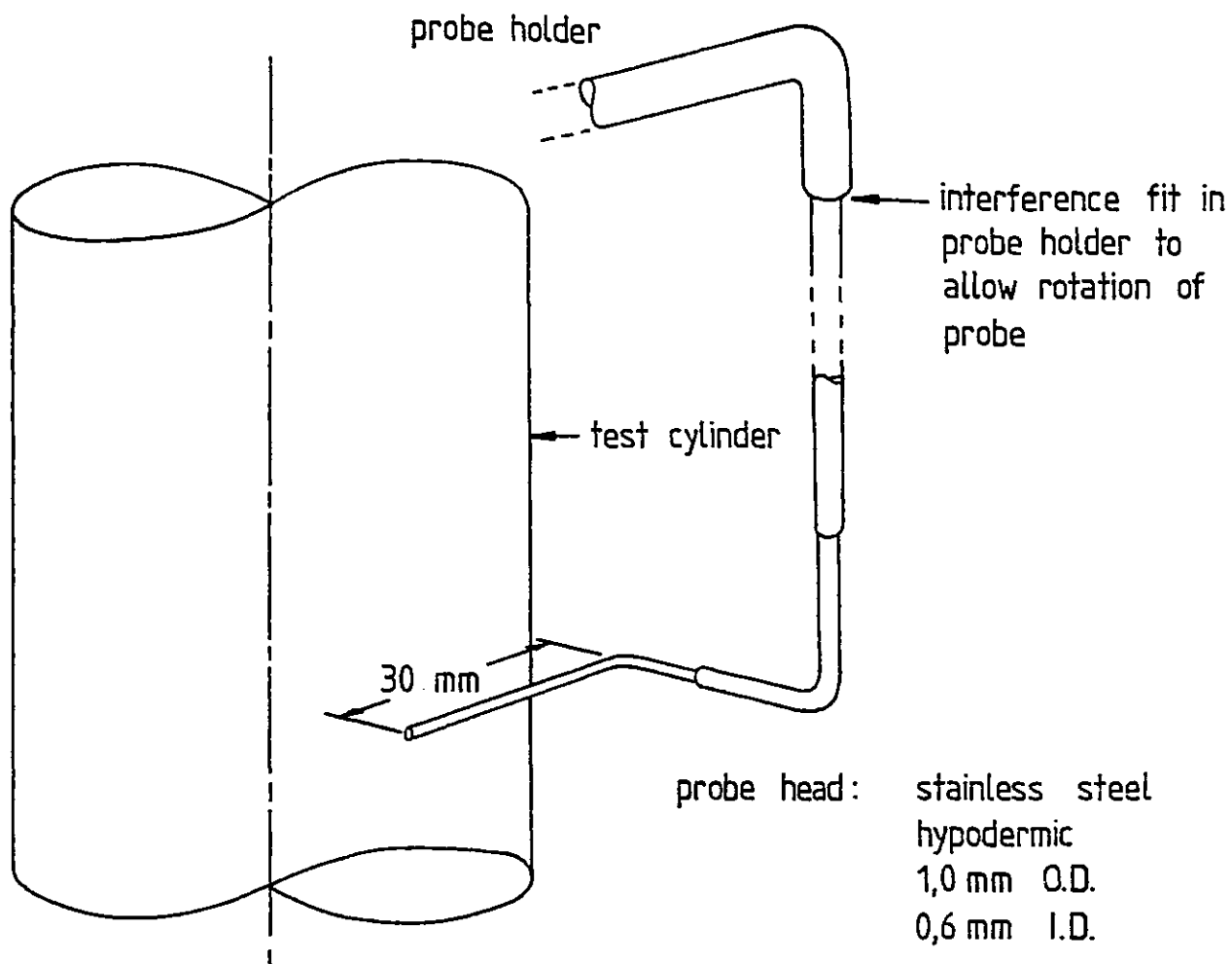


Fig. 2.4 (a) Pitot/Tracer gas sampling probe

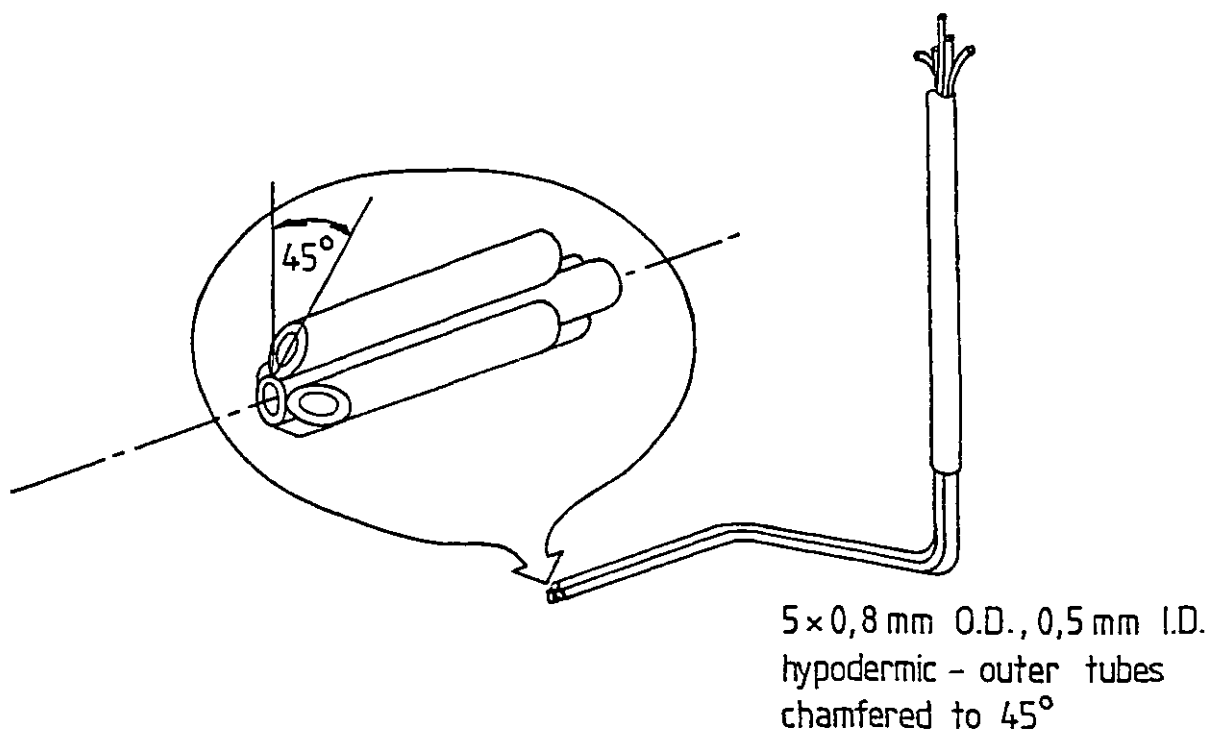


Fig. 2.4(b) Five-hole probe

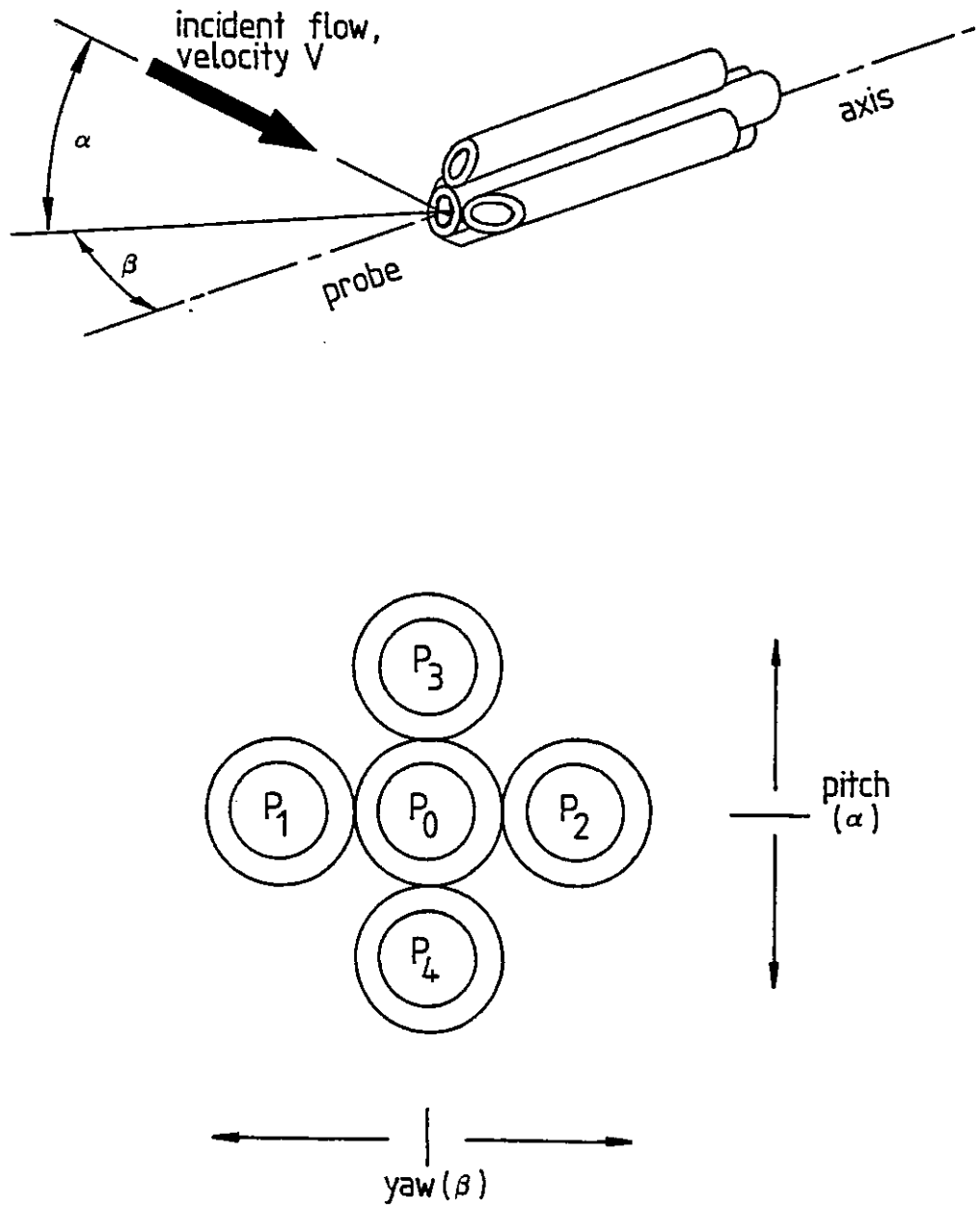


Fig. 2.5 Definition sketch for five-hole probe measurements

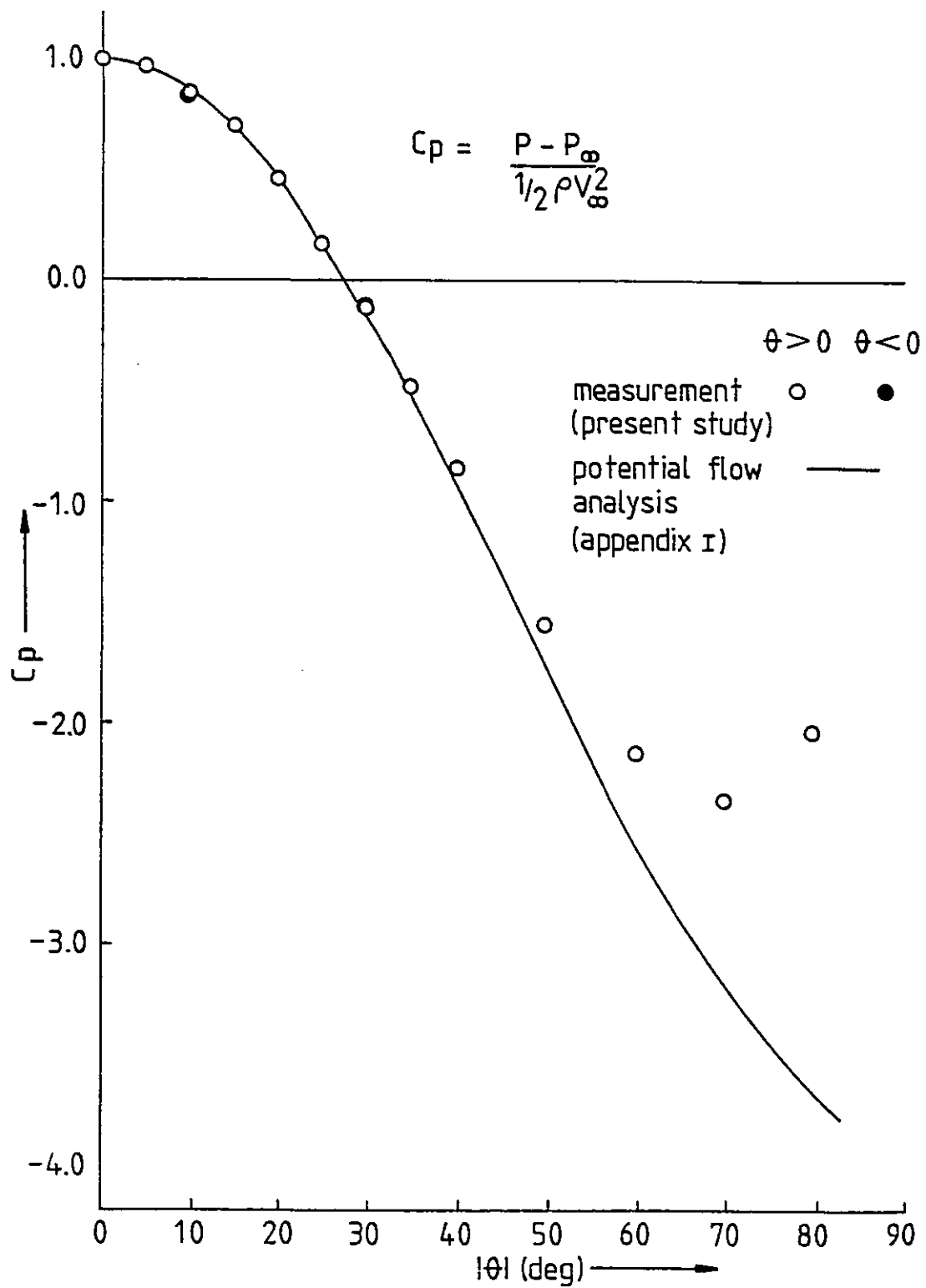


Fig. 3.1 Static pressure distribution on cylinder without ejection

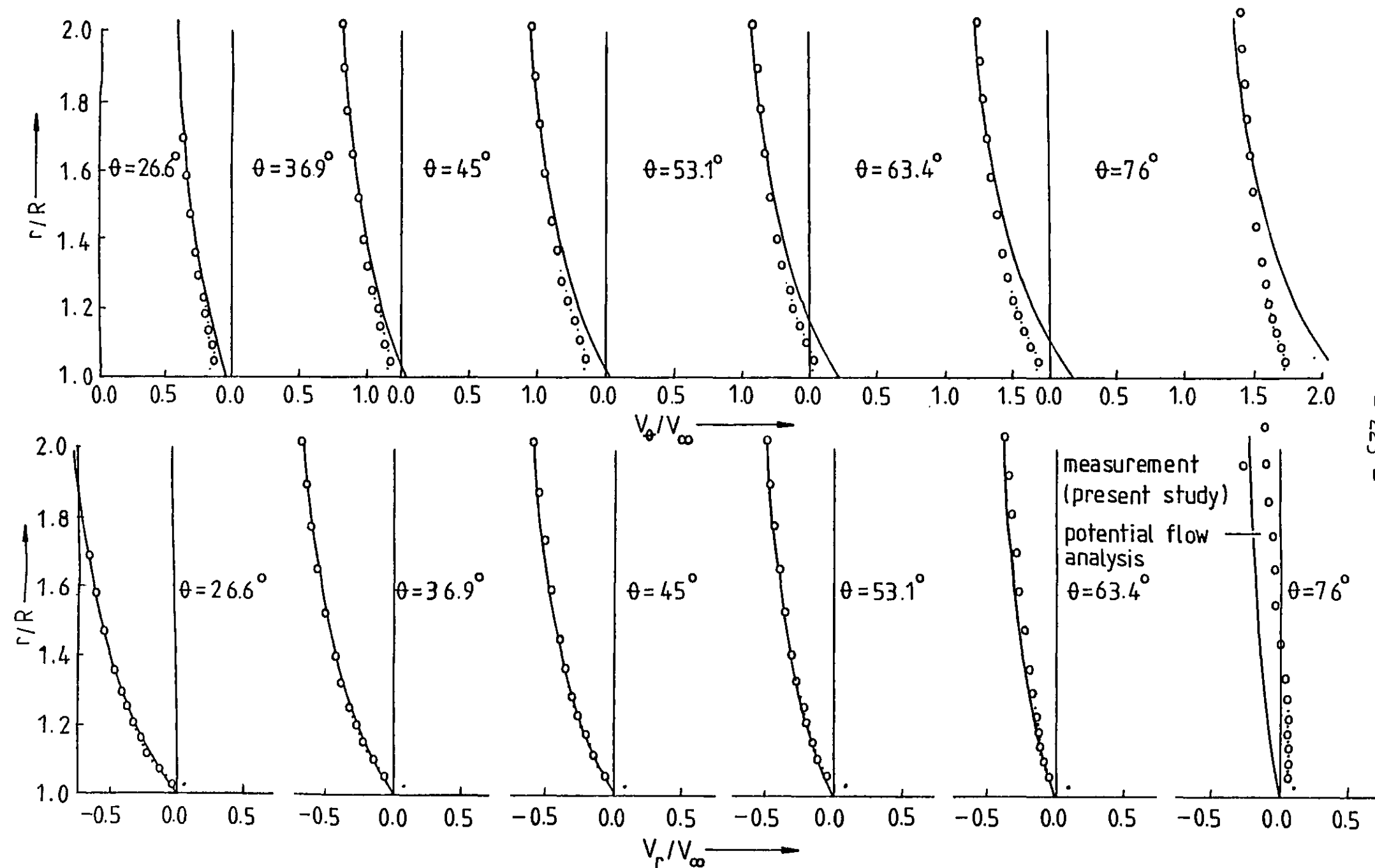


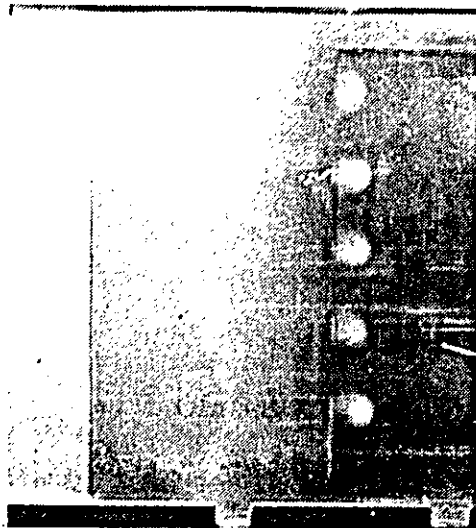
Fig. 3.2 Velocity profiles for cylinder without ejection

Fig. 3.3

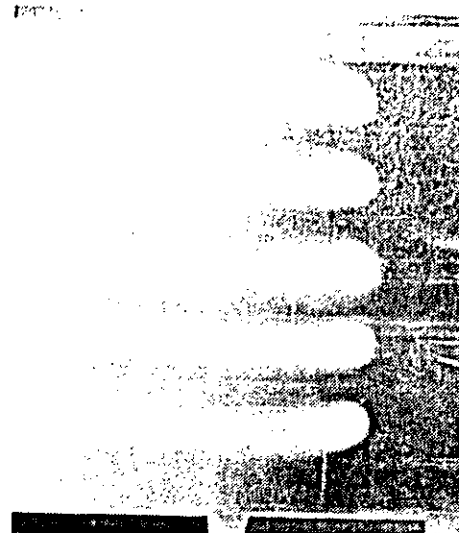
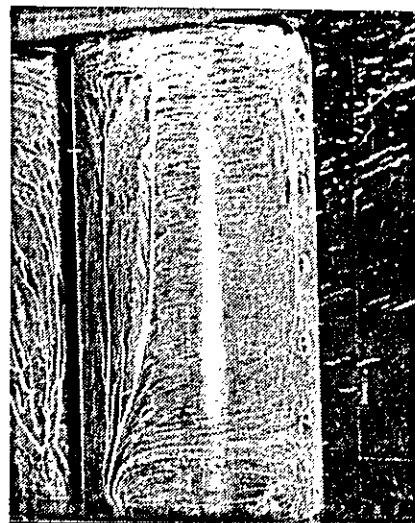
Flow visualisations for $\theta_f = 0^\circ$

BCR 90279

FIG. 3.3



a) $J > 1$ Jets flowing over far surface



c) $J = 0.9$ Symmetrical flow



d) $J = 0.9$ Jets pairing



e) $J = 0.5$

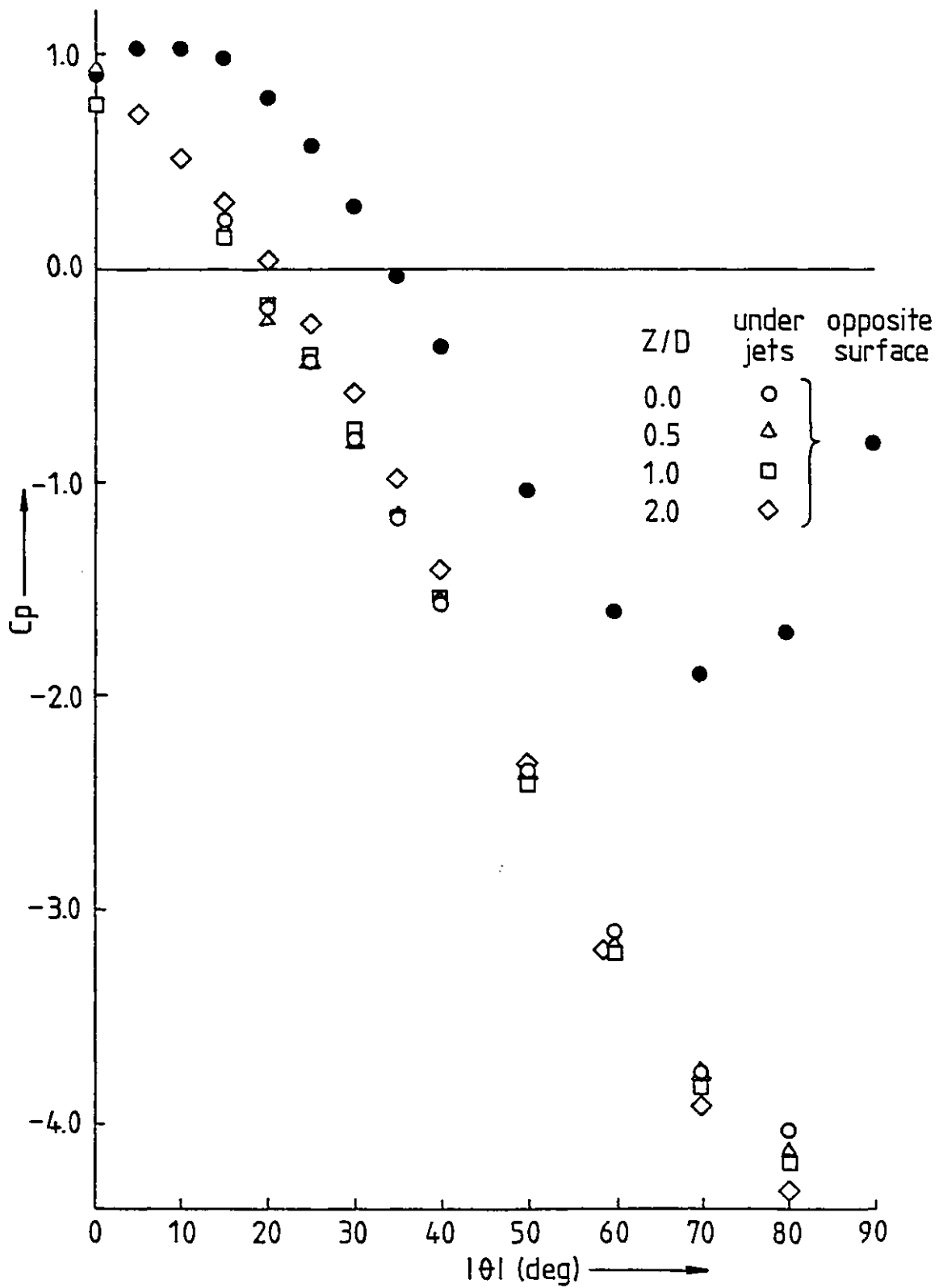


Fig. 3.4 Static pressure distribution, $\Theta_s = 0^\circ$, $J = 1.98$

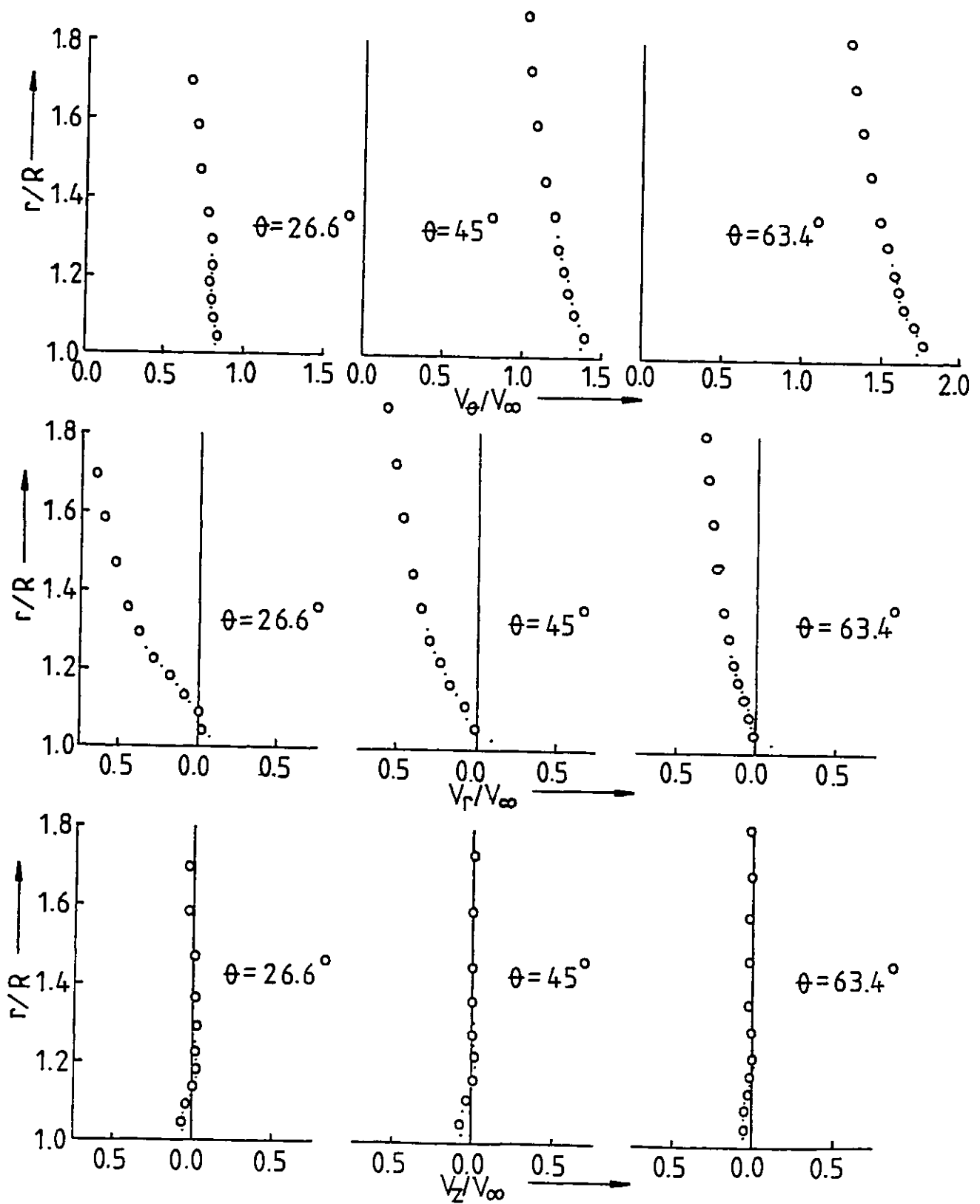


Fig. 3.5(a) Velocity profiles, $\Theta_s = 0^\circ$, $J = 0.89$, $Z/D = 0.0$

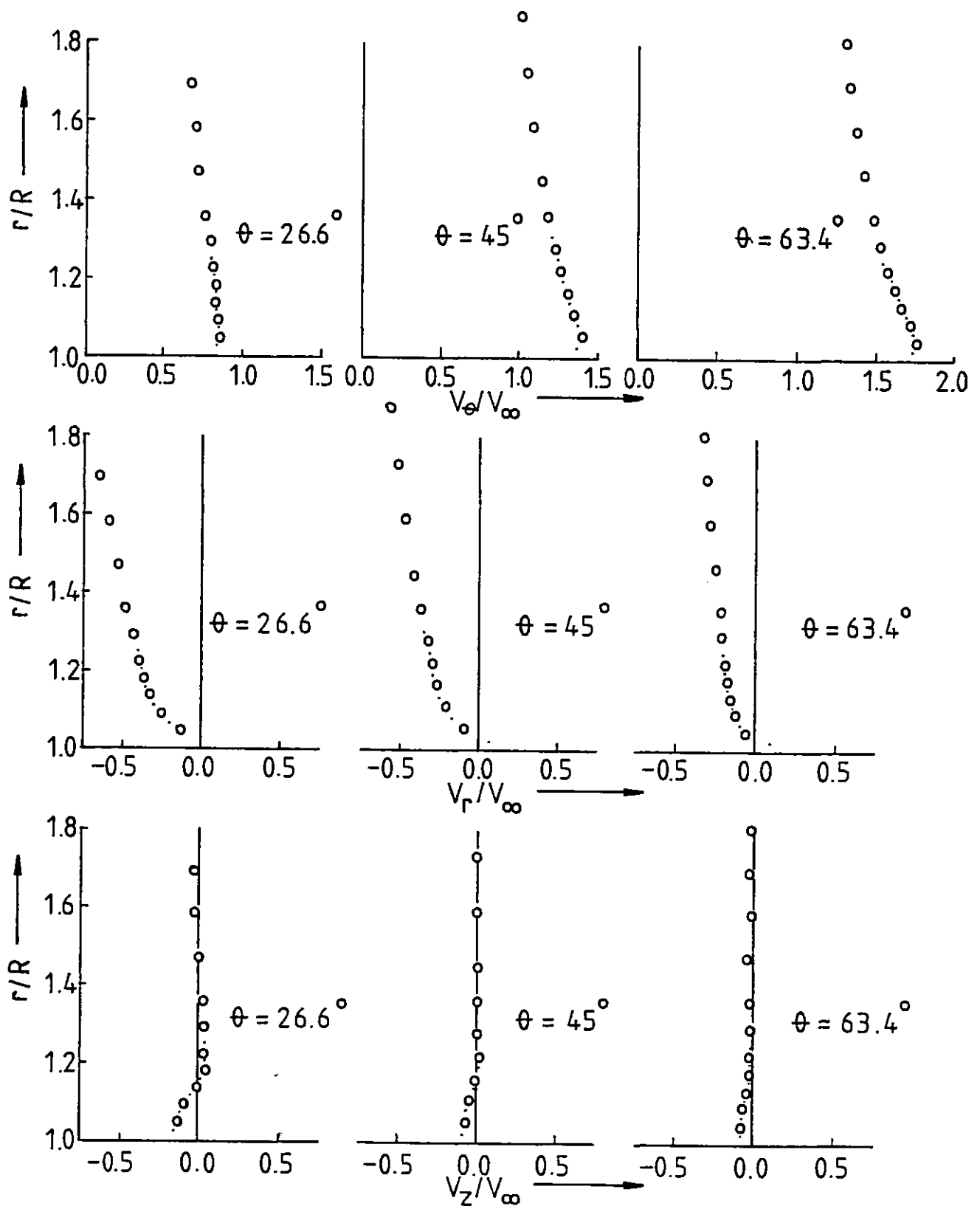


Fig. 3.5(b) Velocity profiles, $\theta_s = 0^\circ$, $J = 0.89$, $z/D = 1.0$

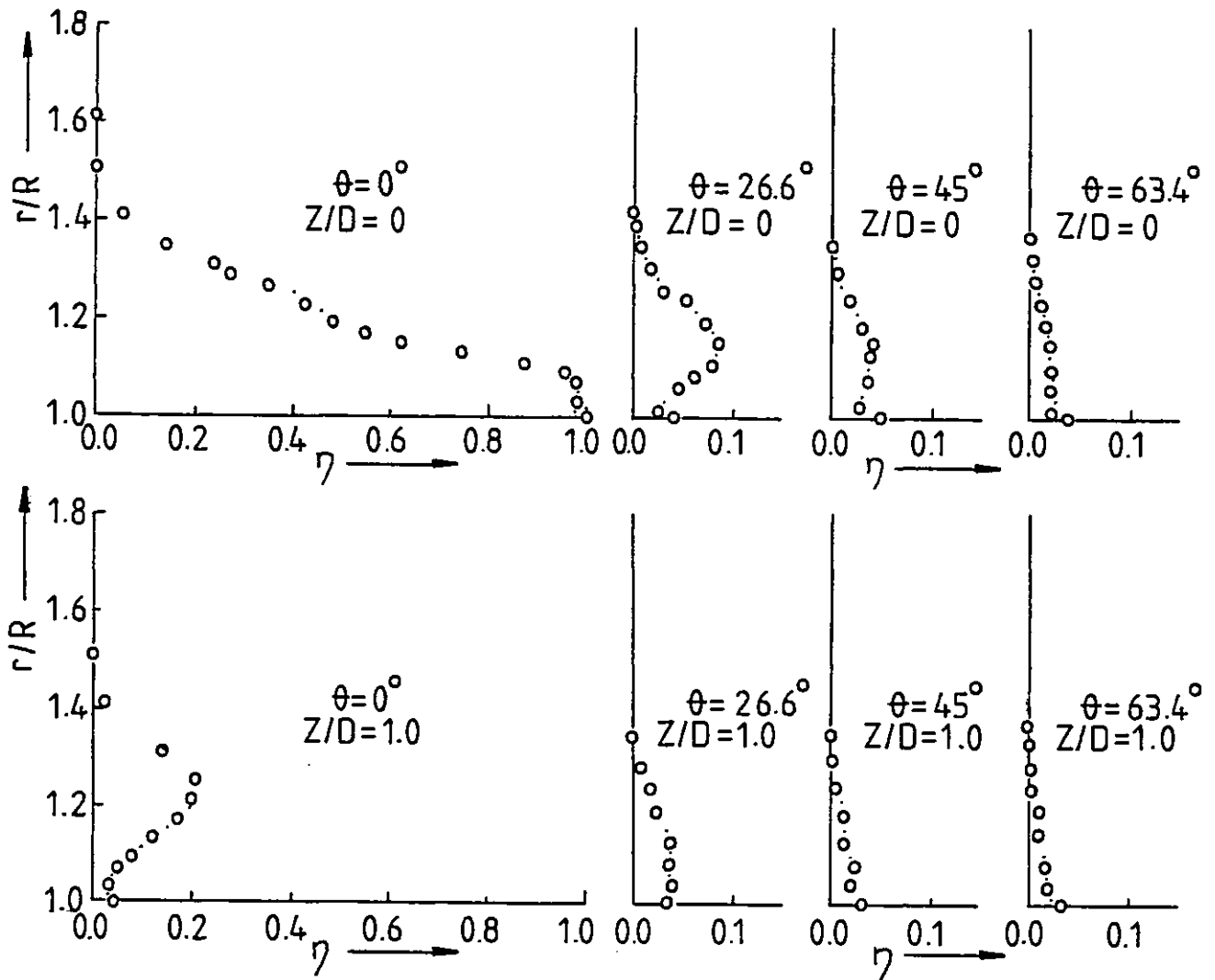


Fig. 3.6 Concentration profiles, $\Theta_\infty = 0^\circ$, $J = 0.89$

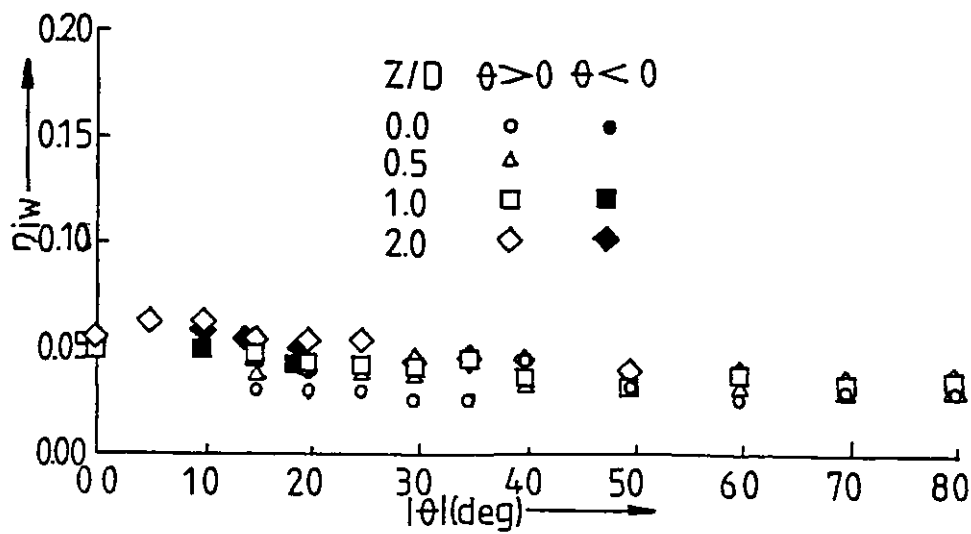


Fig. 3.7 Surface concentration, $\Theta_\infty = 0^\circ$, $J = 0.99$

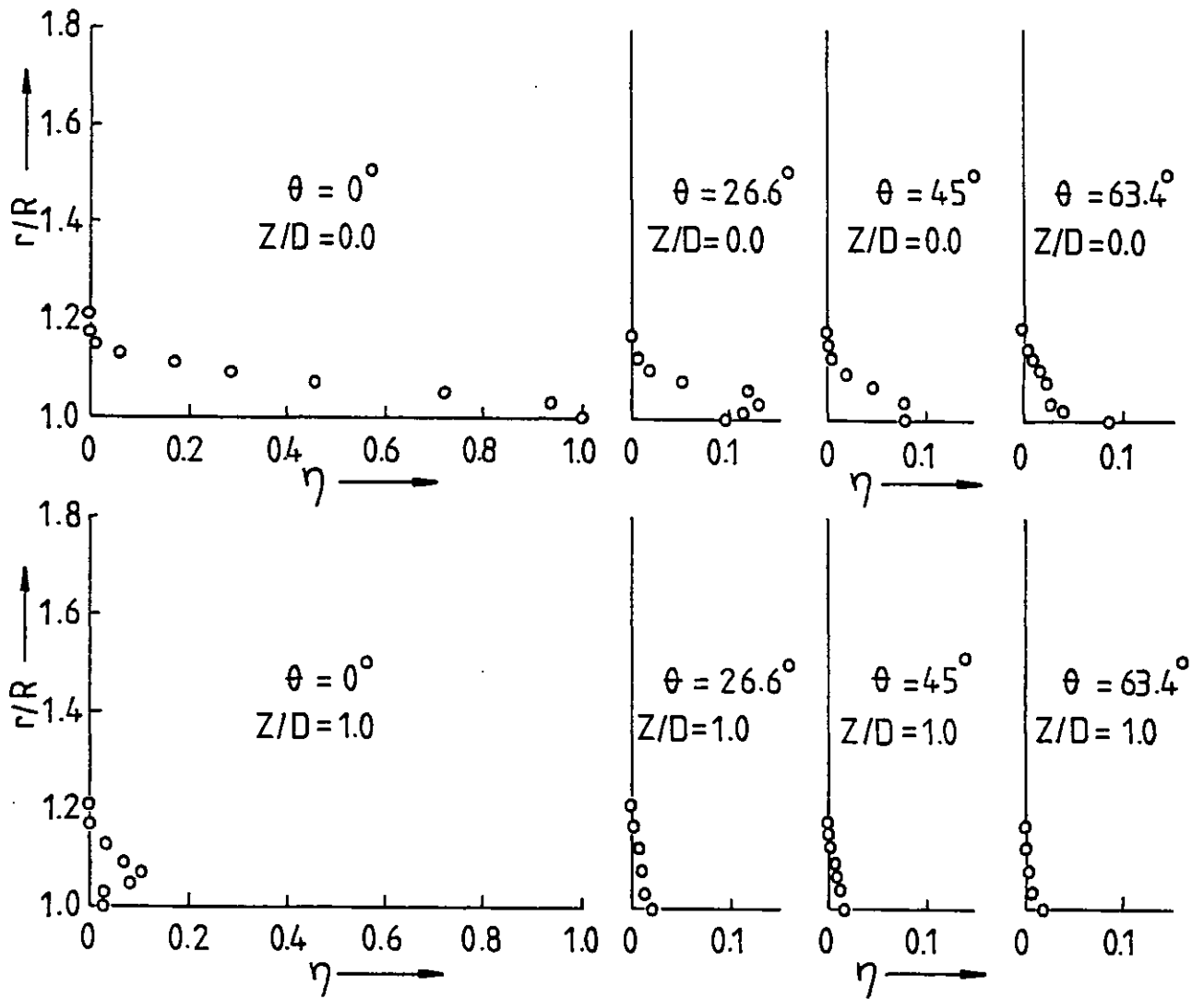


Fig. 3.8 Concentration profiles, $\Theta_r = 0^\circ$, $J = 0.48$

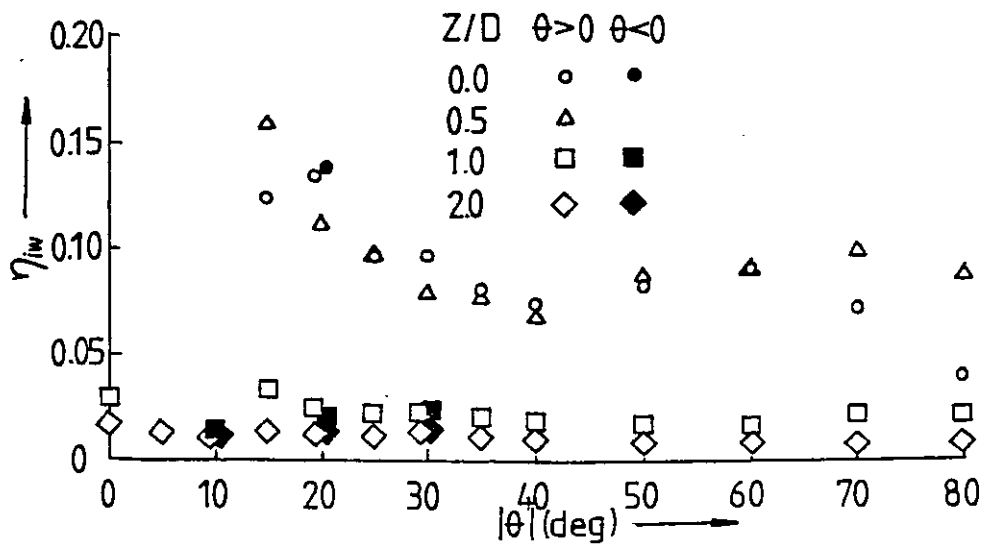
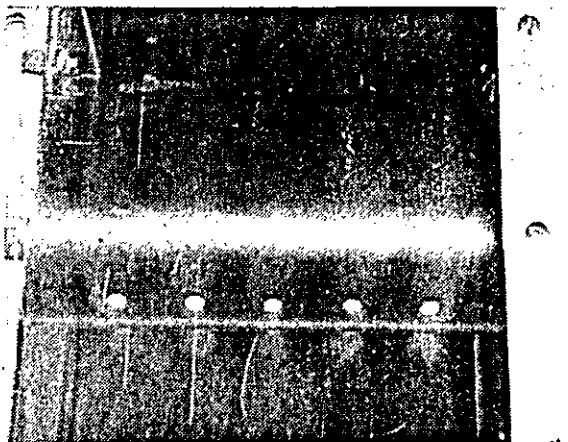
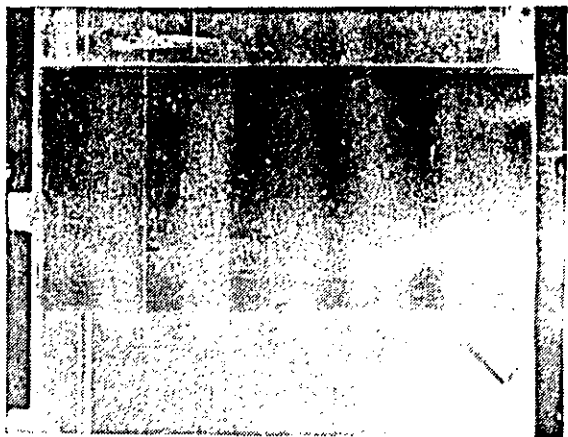
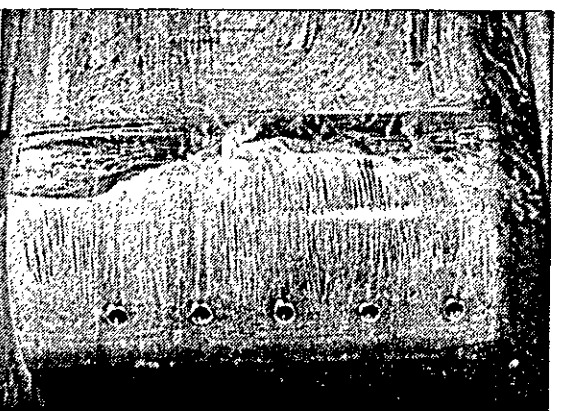
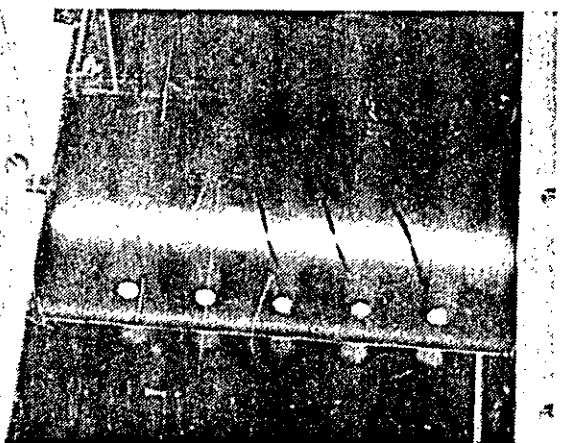
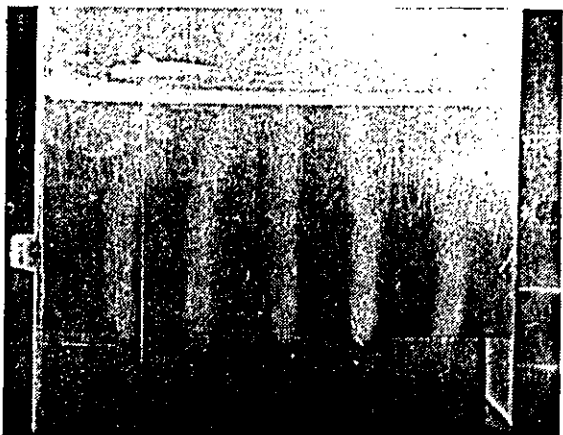


Fig. 3.9 Surface concentration, $\Theta_r = 0^\circ$, $J = 0.48$

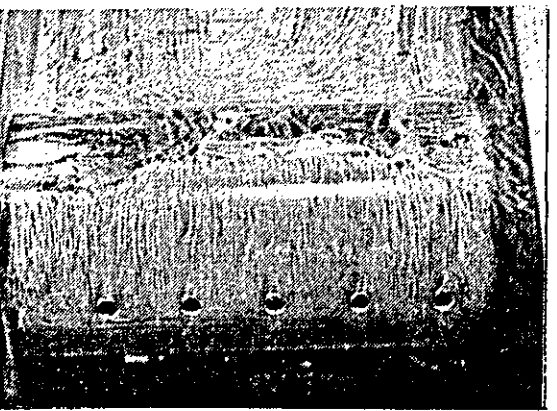
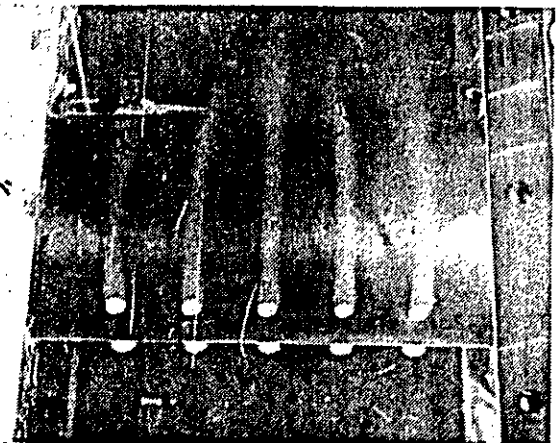
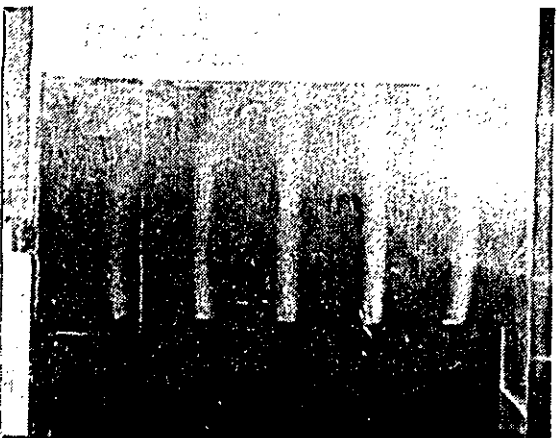
No.



a) $J = 2.0$



b) $J = 1.0$



c) $J = 0.5$

Fig. 3.10 Flow visualisations for $\Theta_r = \pm 20^\circ$

BCR 90279

FIG. 3.10

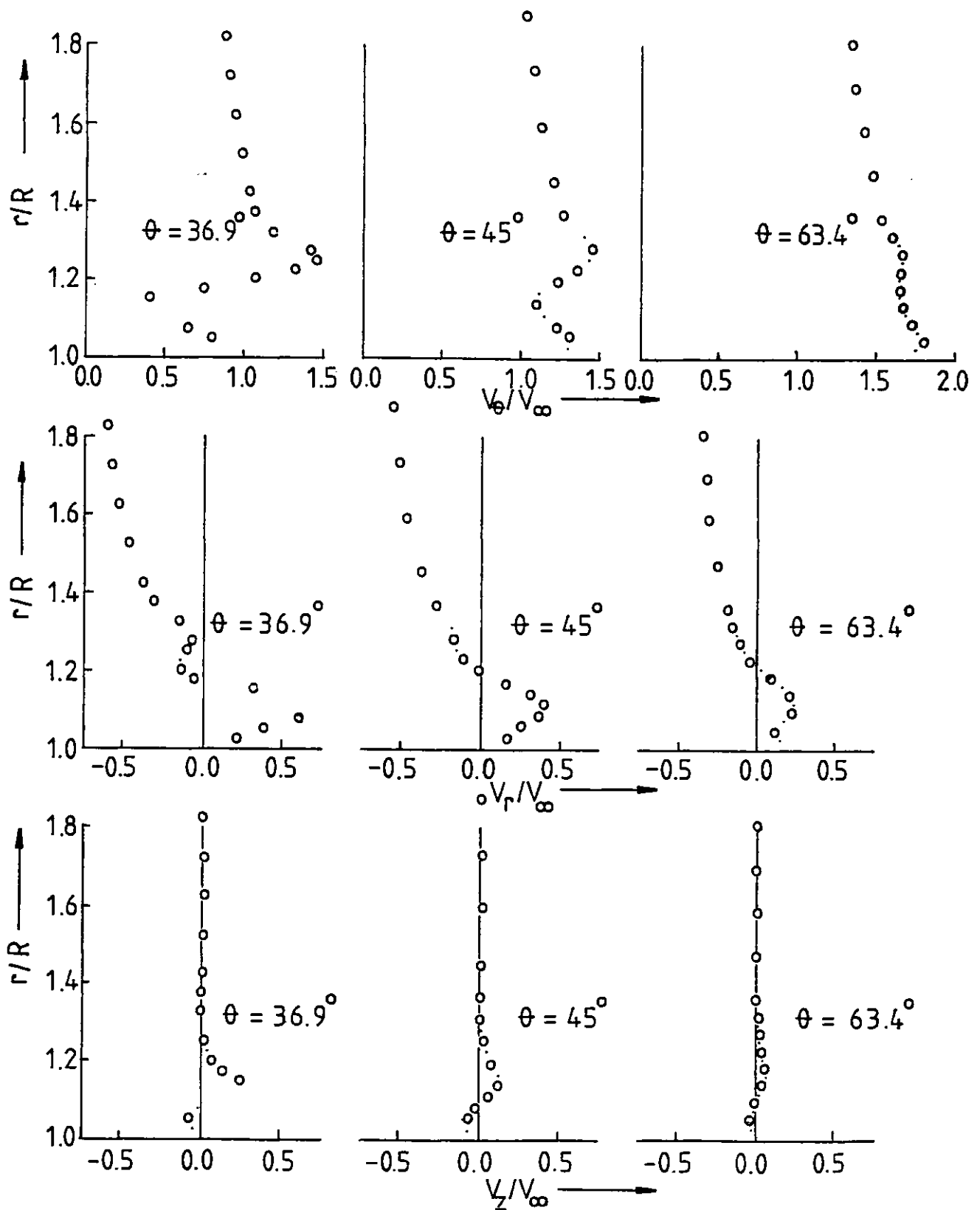


Fig. 3.11(a) Velocity profiles, $\theta_\tau = \pm 20^\circ$, $J = 1.98$, $Z/D = 0.0$

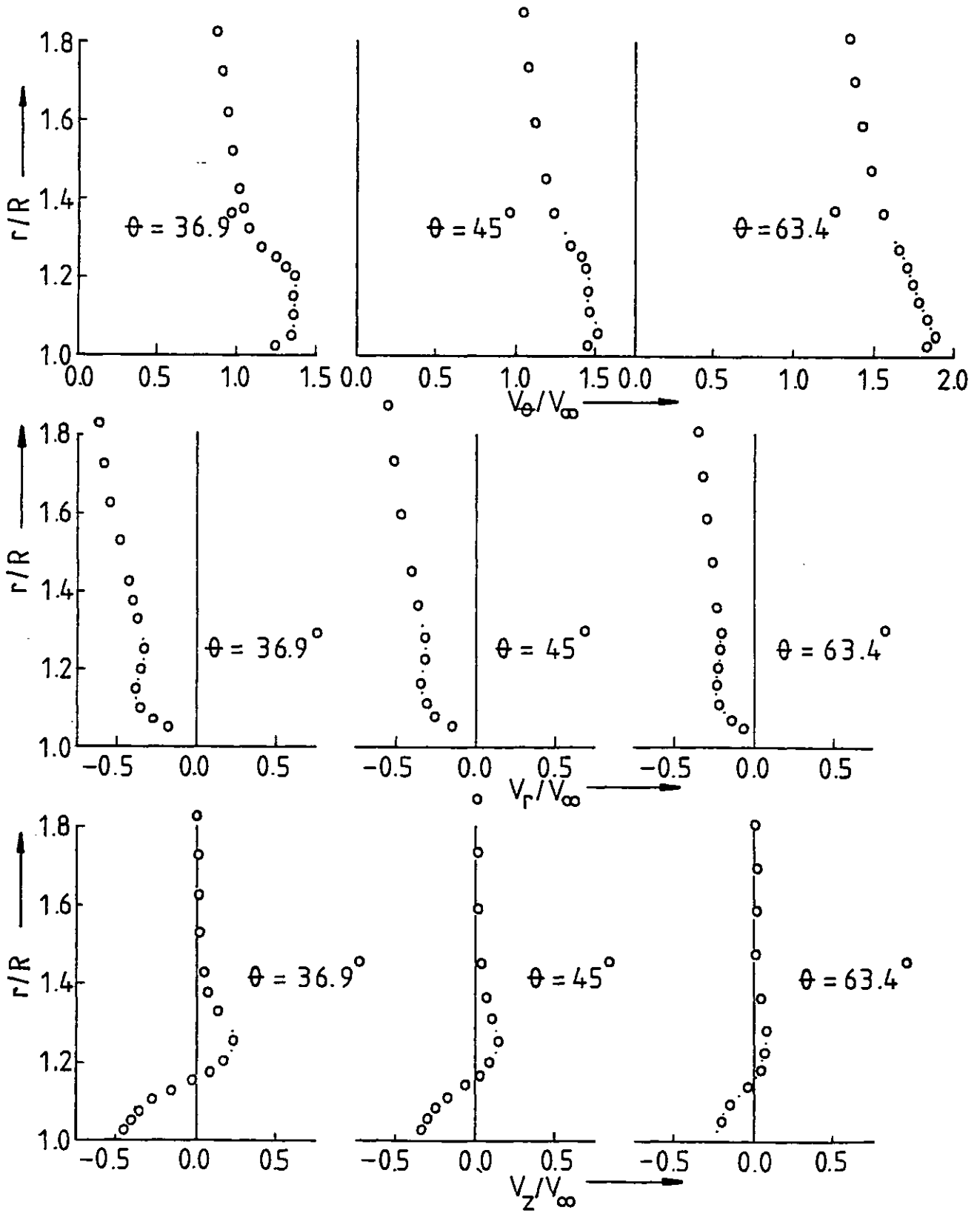


Fig. 3.11(b) Velocity profiles, $\Theta_s = \pm 20^\circ$, $J = 1.98$, $Z/D = 1.0$

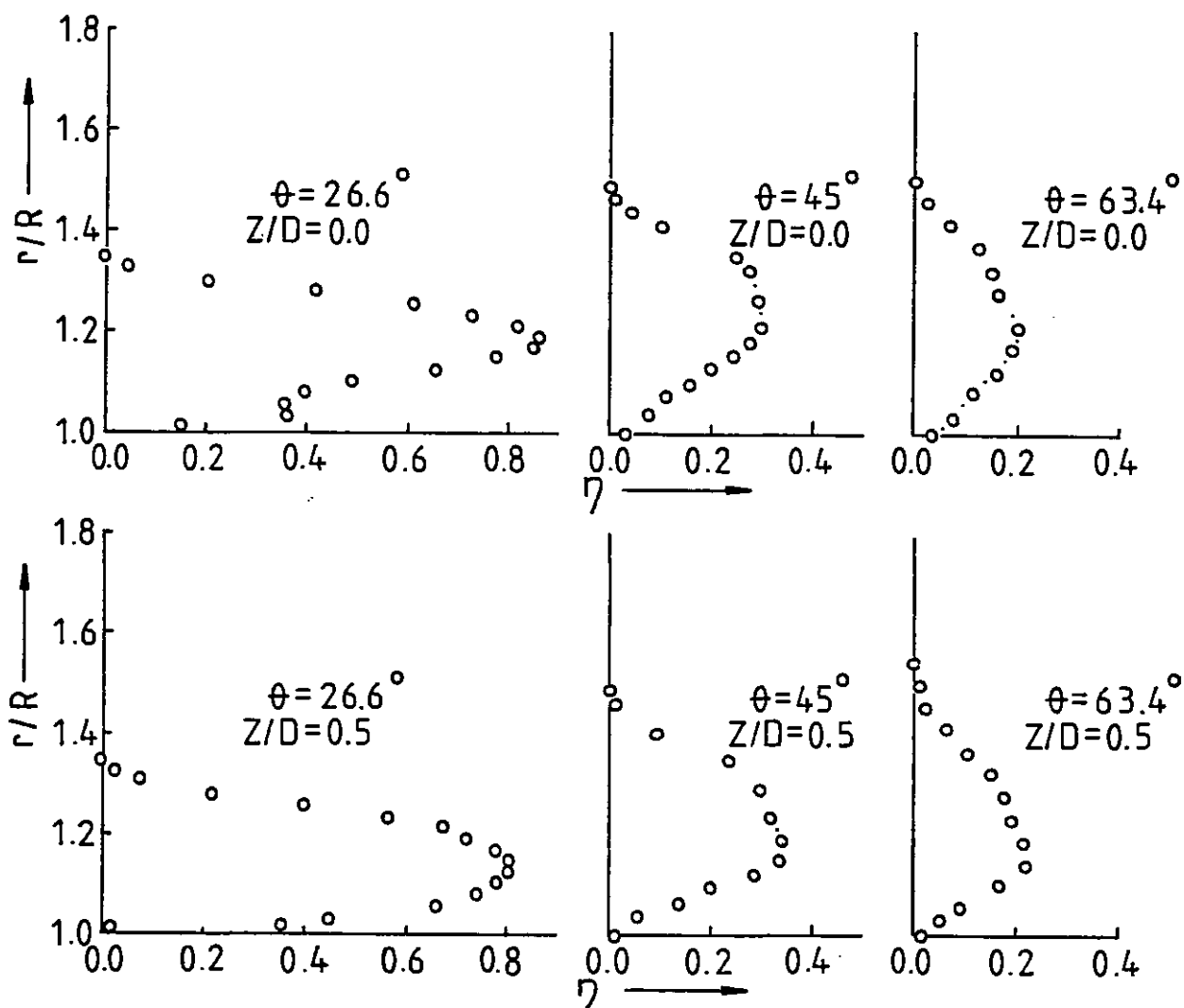


Fig. 3.12. Concentration profiles, $\Theta_x = \pm 20^\circ$, $J = 1.98$

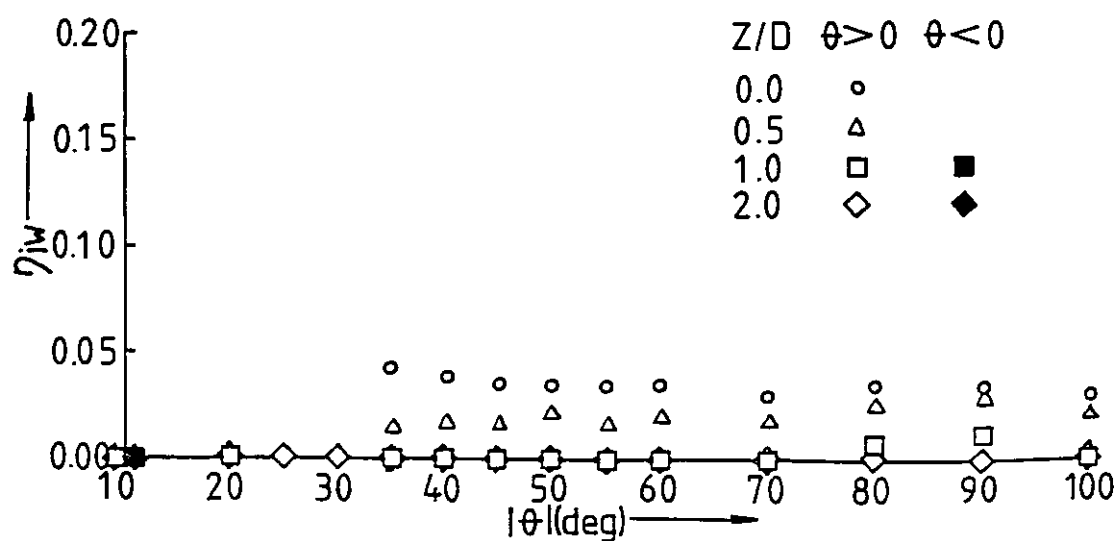


Fig. 3.13. Surface concentration, $\Theta_x = \pm 20^\circ$, $J = 1.98$

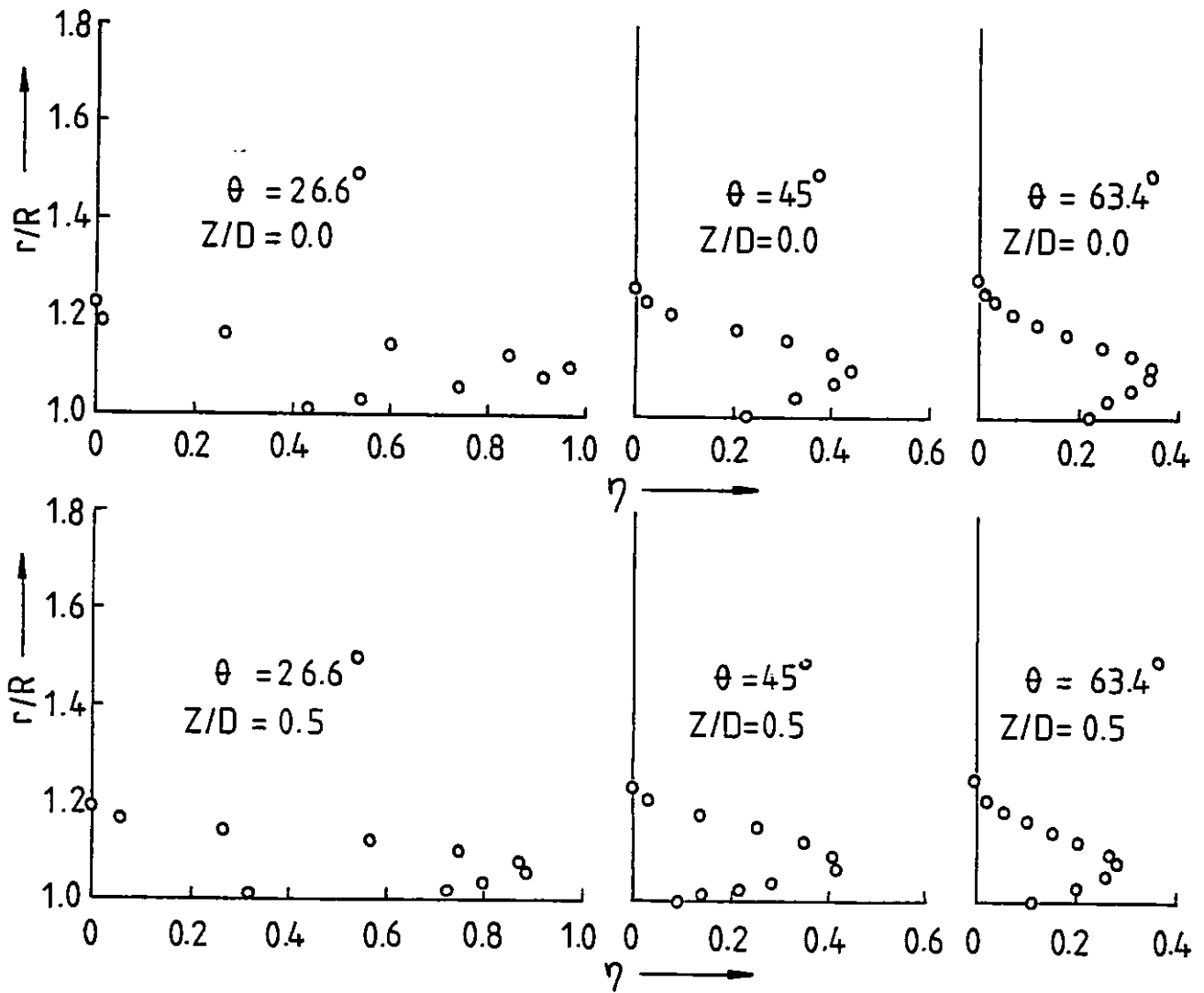


Fig. 3.14 Concentration profiles, $\Theta_x = \pm 20^\circ$, $J = 1.11$

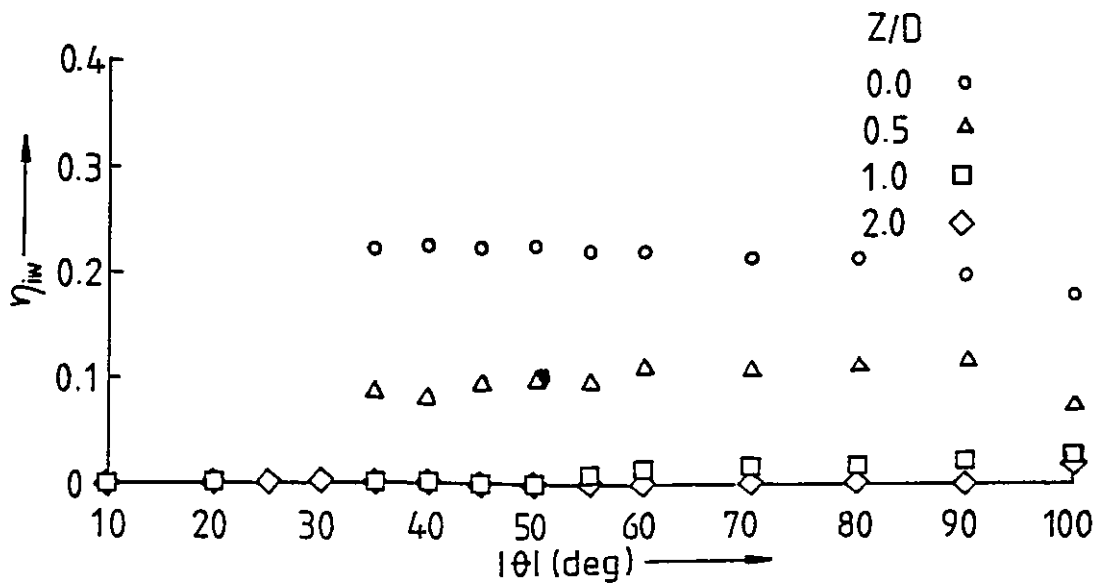


Fig. 3.15 Surface concentration, $\Theta_x = \pm 20^\circ$, $J = 1.11$

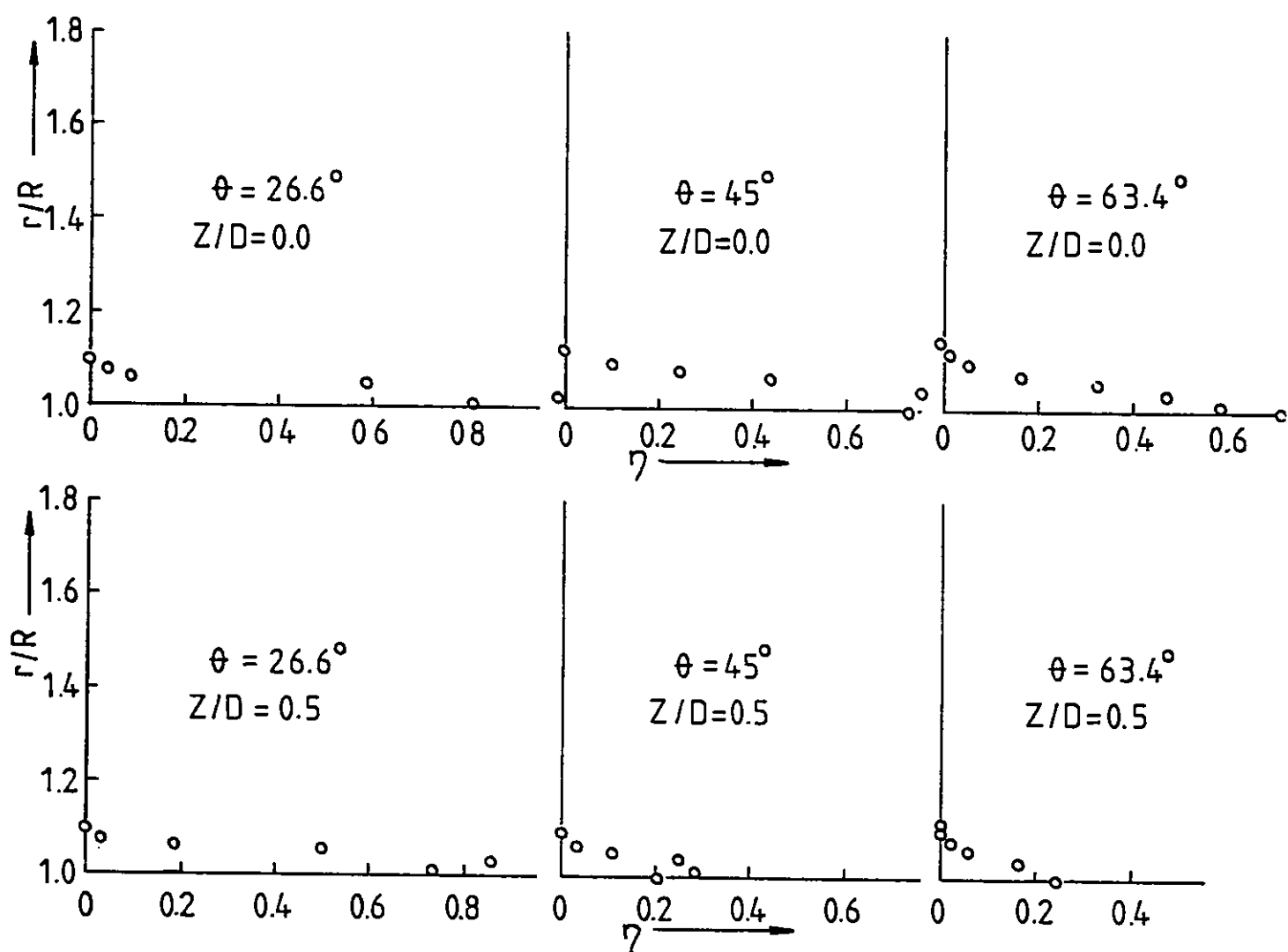


Fig. 3.16 Concentration profiles, $\Theta_x = \pm 20^\circ$, $J = 0.55$

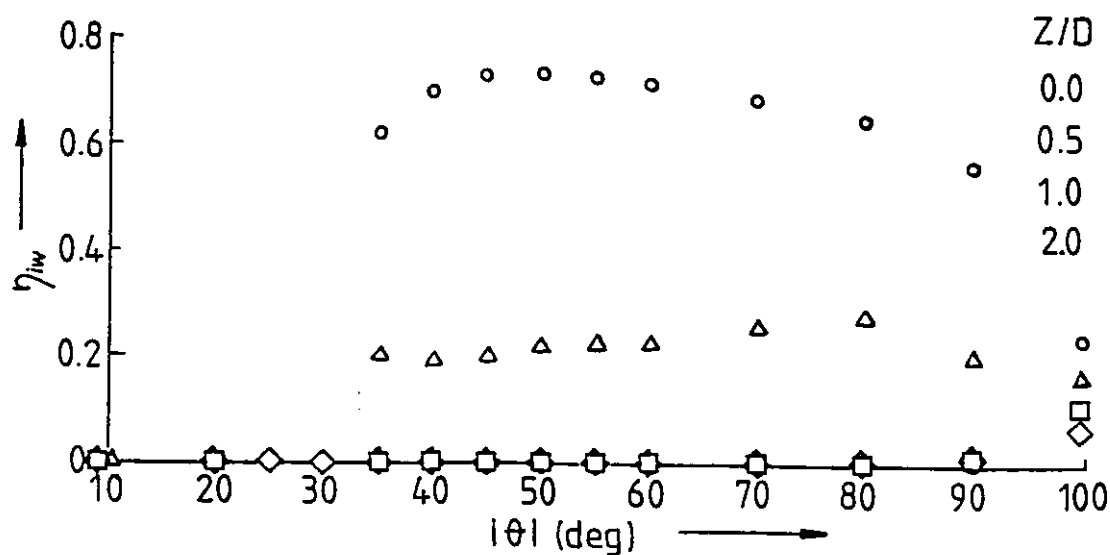
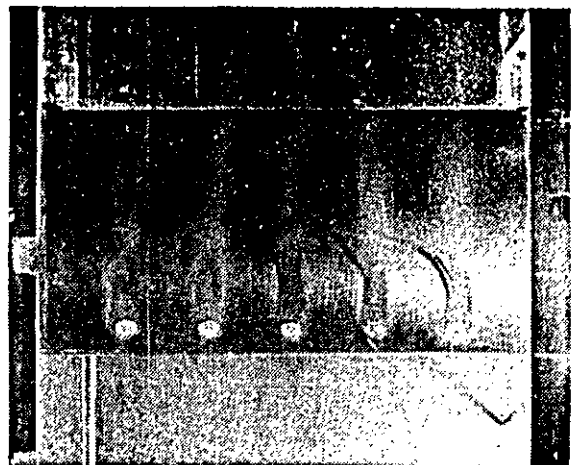
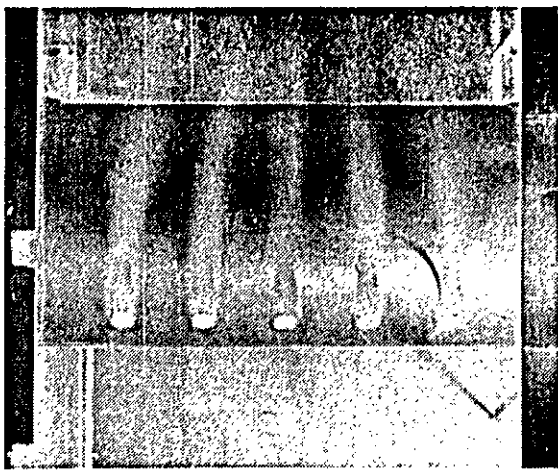
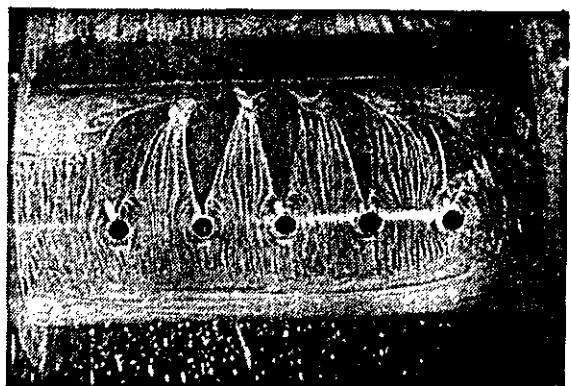


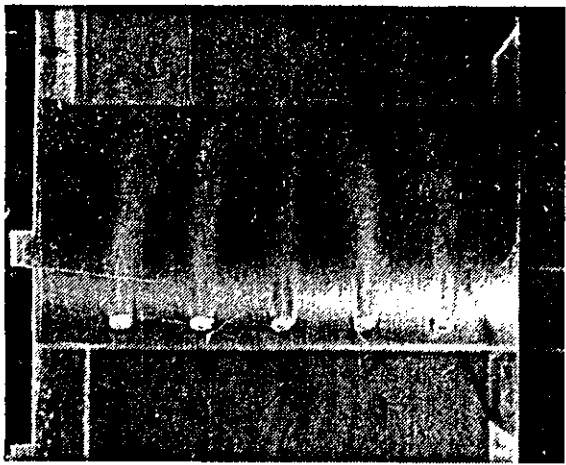
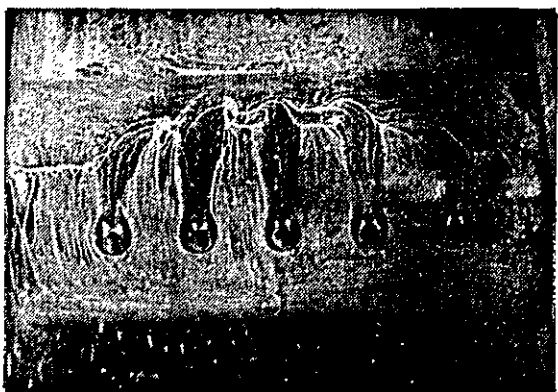
Fig. 3.17 Surface concentration, $\Theta_x = \pm 20^\circ$, $J = 0.55$



a) $J = 1.8$



b) $J = 1.0$



c) $J = 0.5$

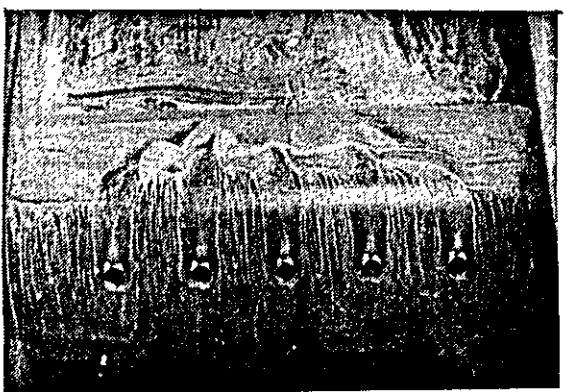


Fig. 3.18

Flow visualisations for $\theta_r = \pm 40^\circ$

RCR 90279

FIG. 3.18

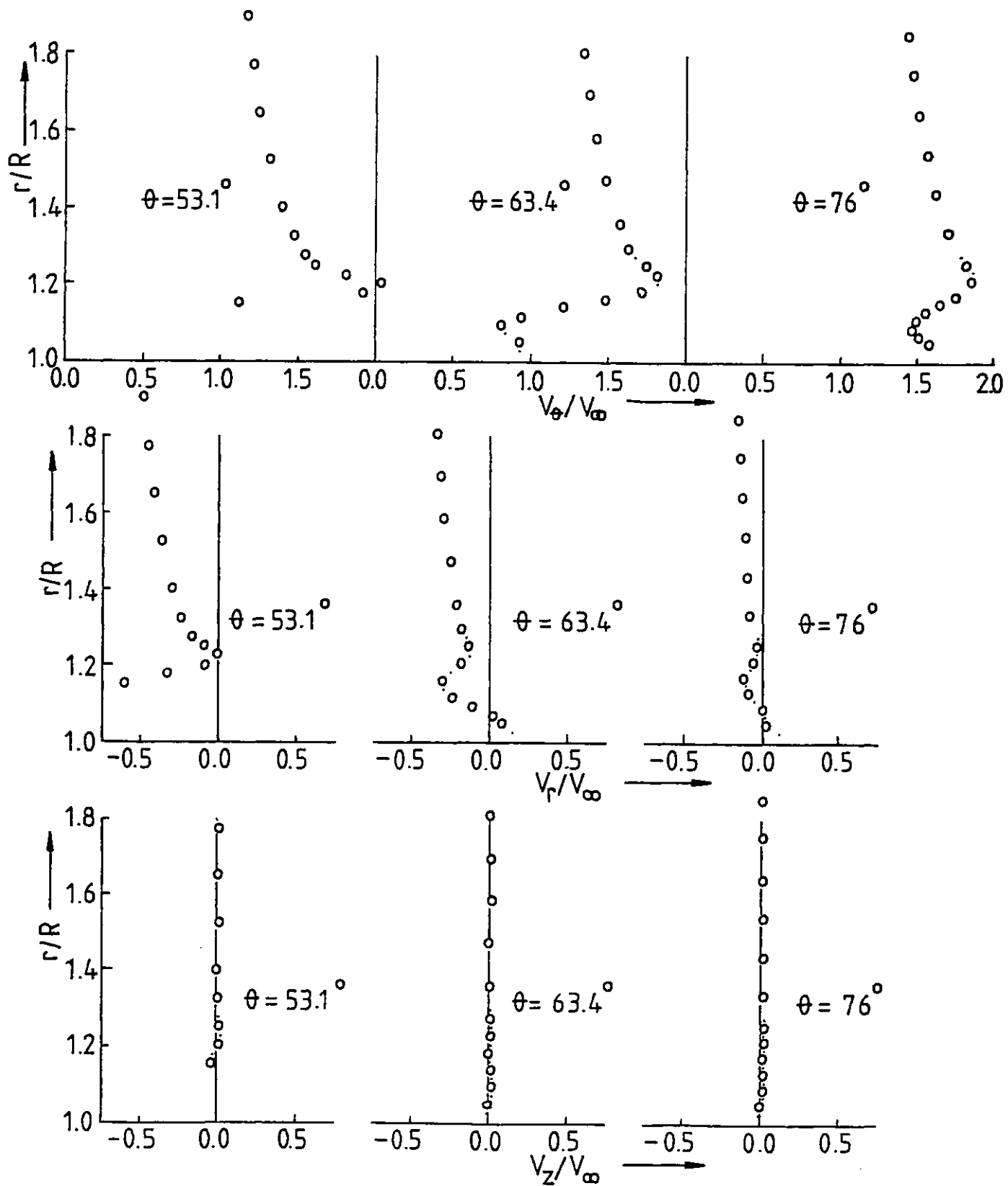


Fig. 3.19(a) Velocity profiles, $\Theta_x = \pm 40^\circ$, $J = 1.81$, $z/D = 0.0$

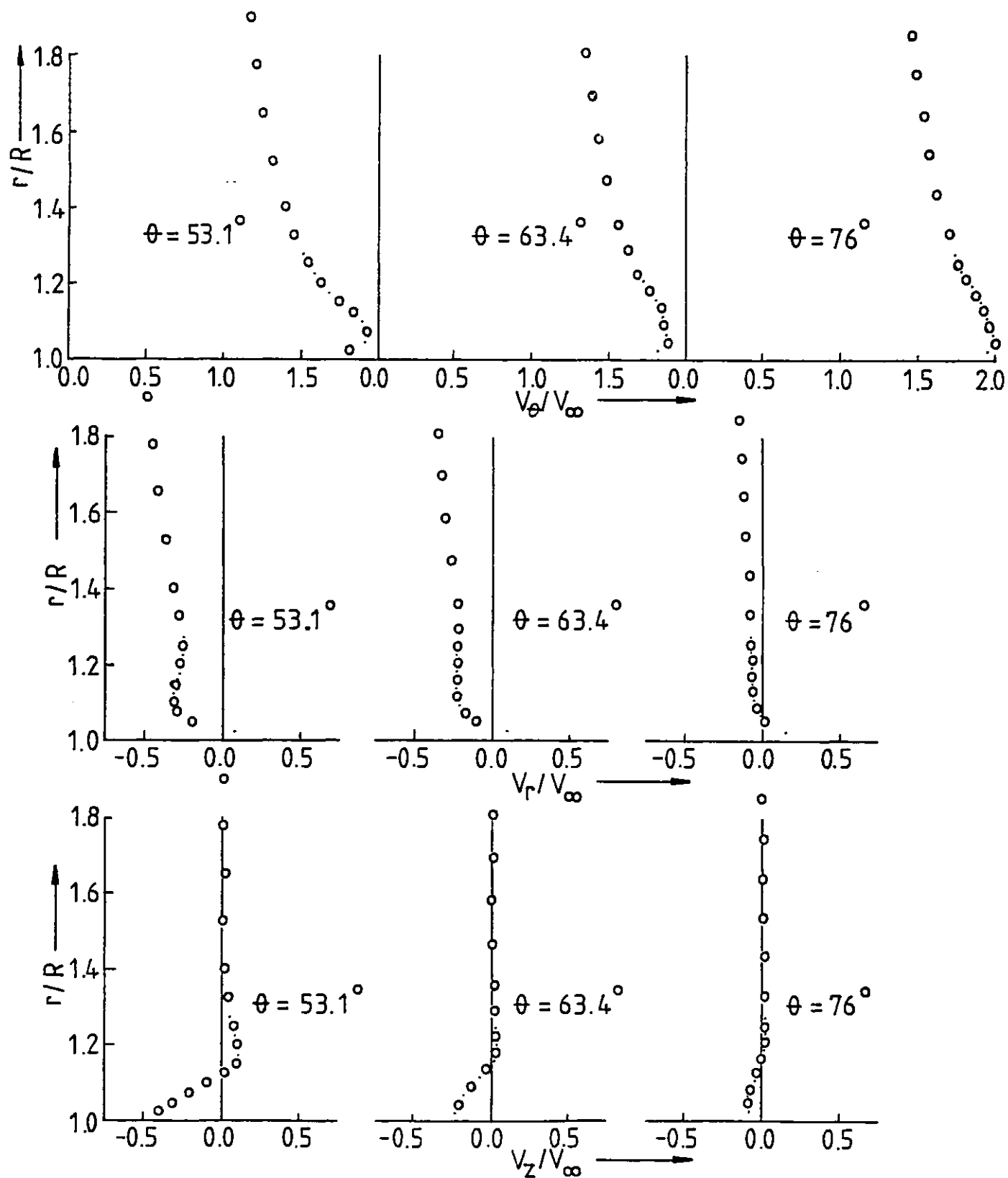


Fig. 3.19(b) Velocity profiles, $\Theta_x = \pm 40^\circ$, $J = 1.81$, $Z/D = 1.0$

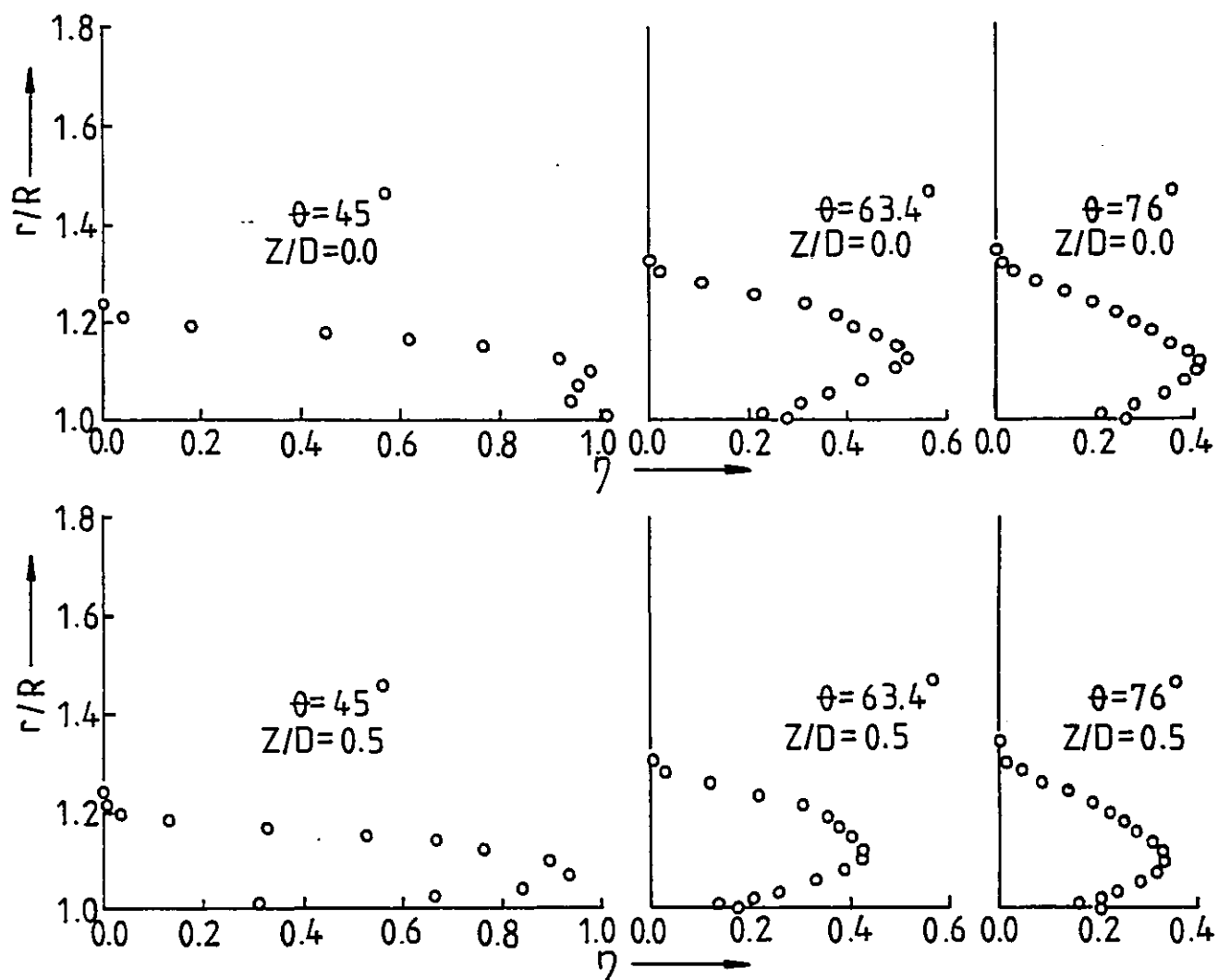


Fig. 3.20 Concentration profiles, $\Theta_r = \pm 40^\circ$, $J = 1.81$

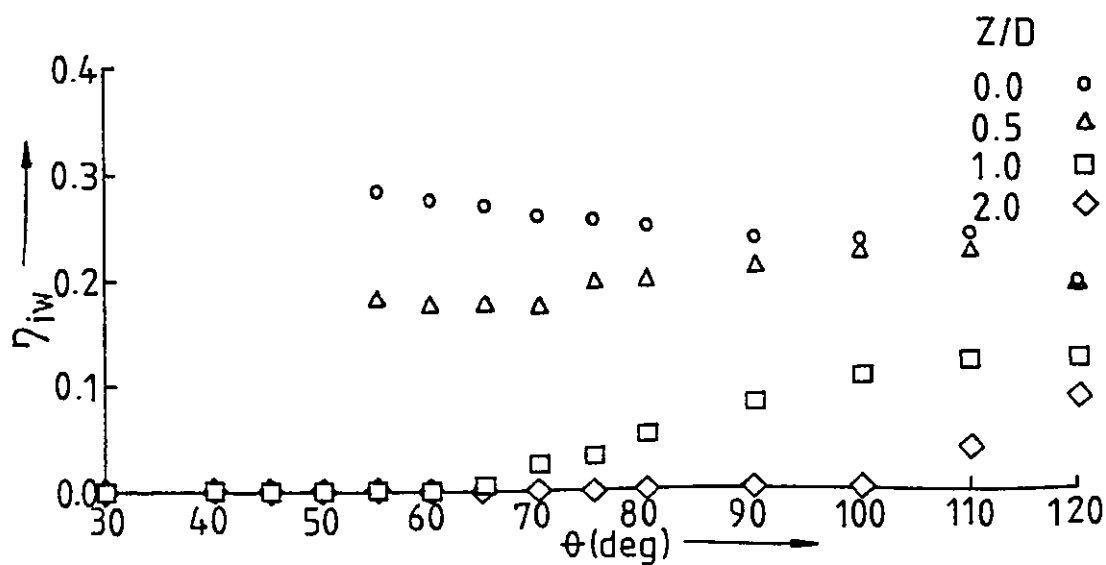


Fig. 3.21 Surface concentration, $\Theta_r = \pm 40^\circ$, $J = 1.81$

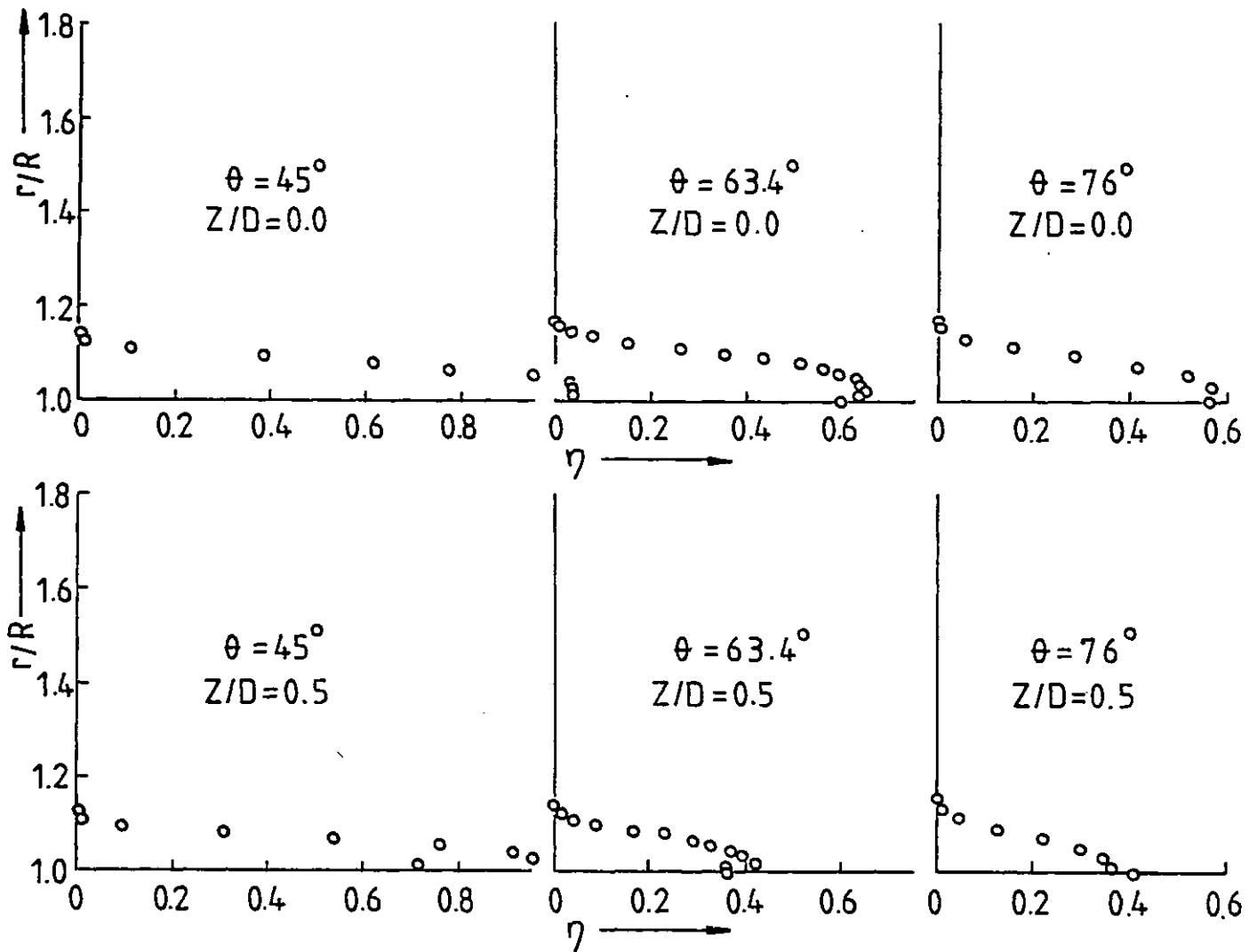


Fig. 3.22 Concentration profiles, $\Theta_{\gamma} = \pm 40^\circ$, $J = 1.0$

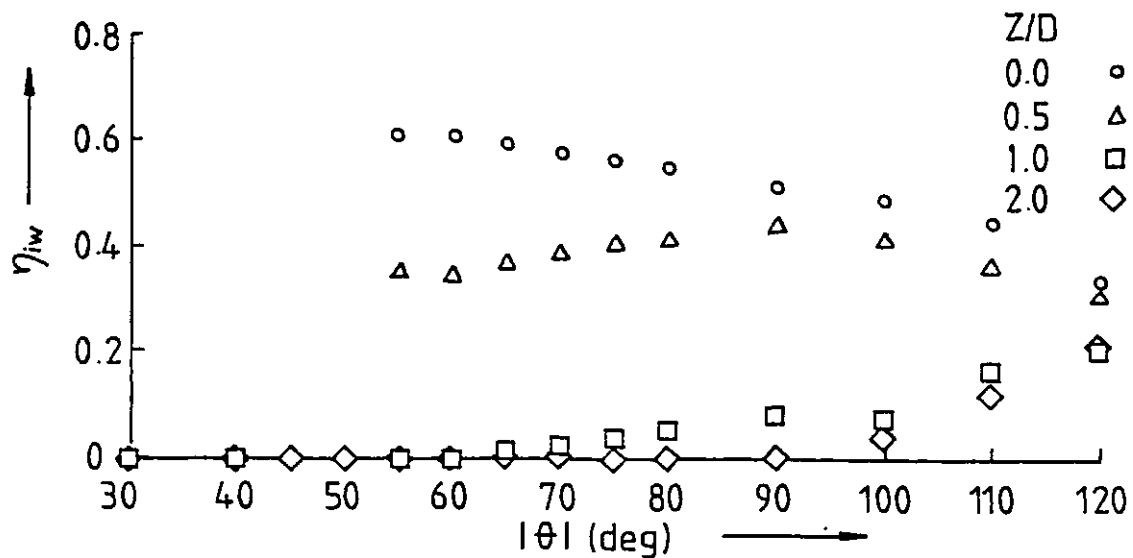


Fig. 3.23 Surface concentration, $\Theta_{\gamma} = \pm 40^\circ$, $J = 1.0$

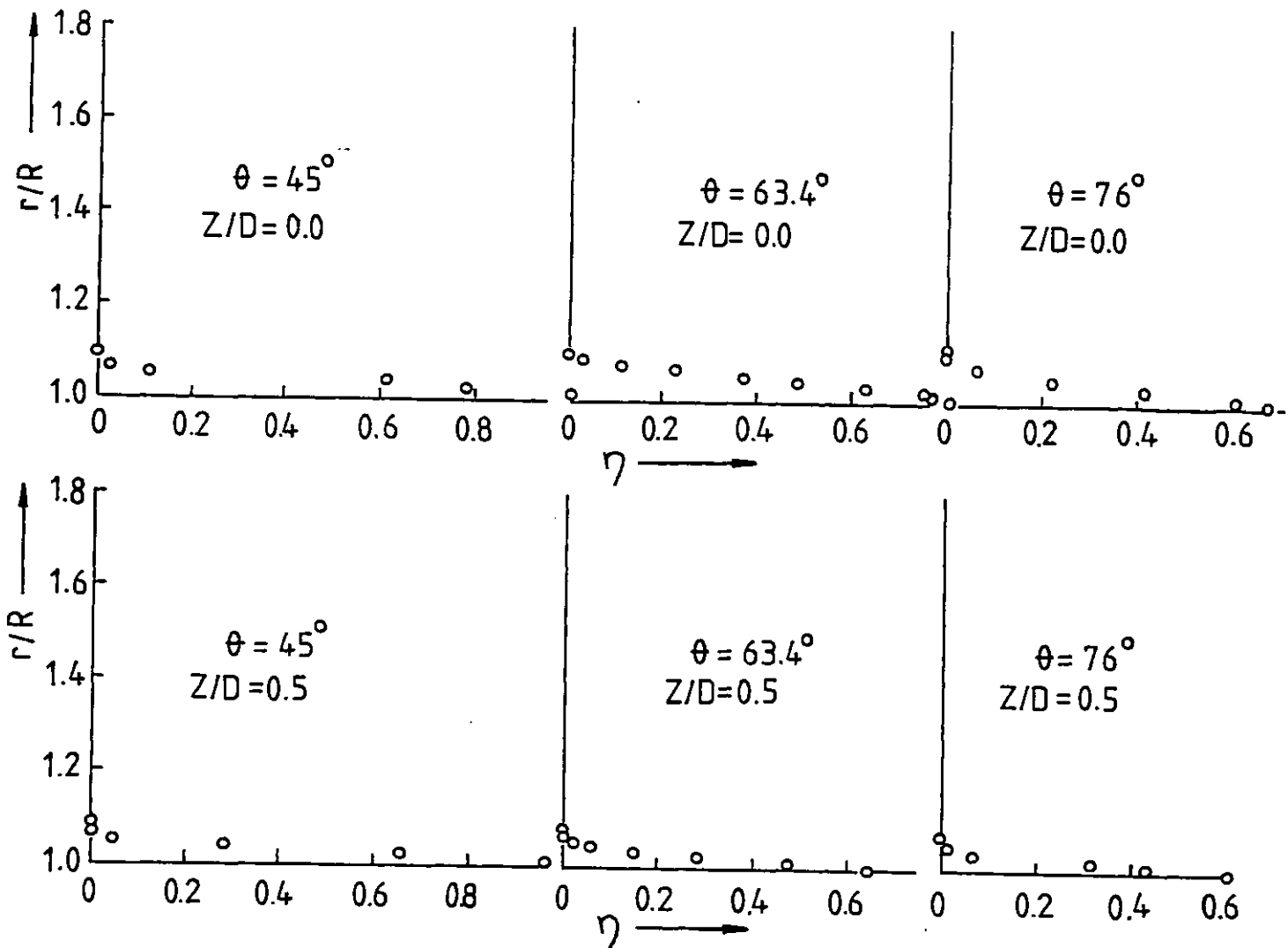


Fig. 3.24 Concentration profiles, $\Theta_{\pm} = \pm 40^\circ$, $J = 0.5$

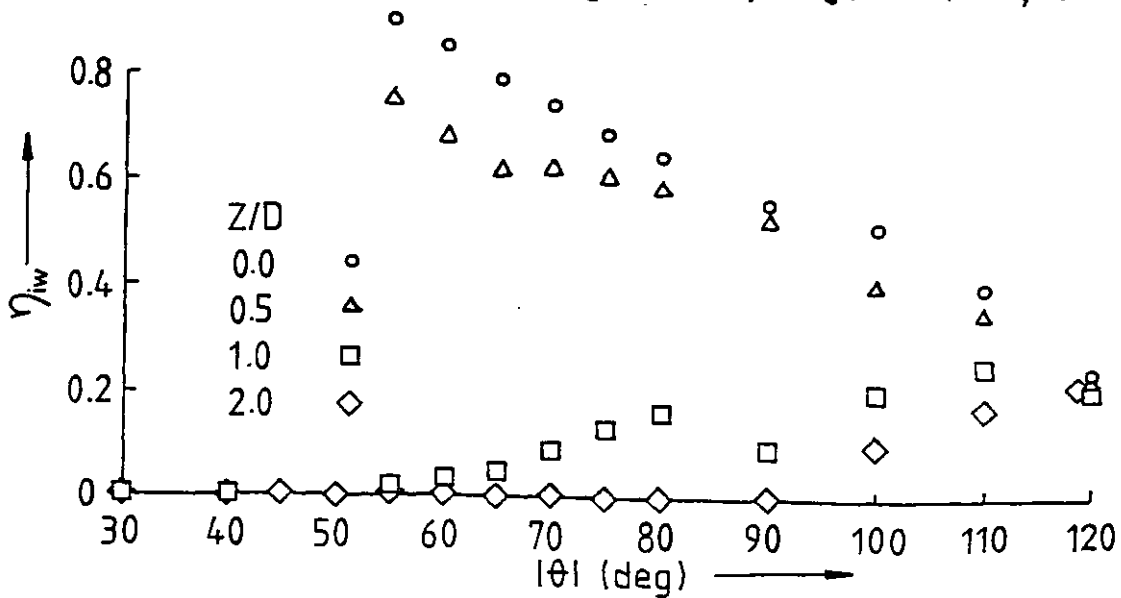


Fig. 3.25 Surface concentration, $\Theta_{\pm} = \pm 40^\circ$, $J = 0.5$

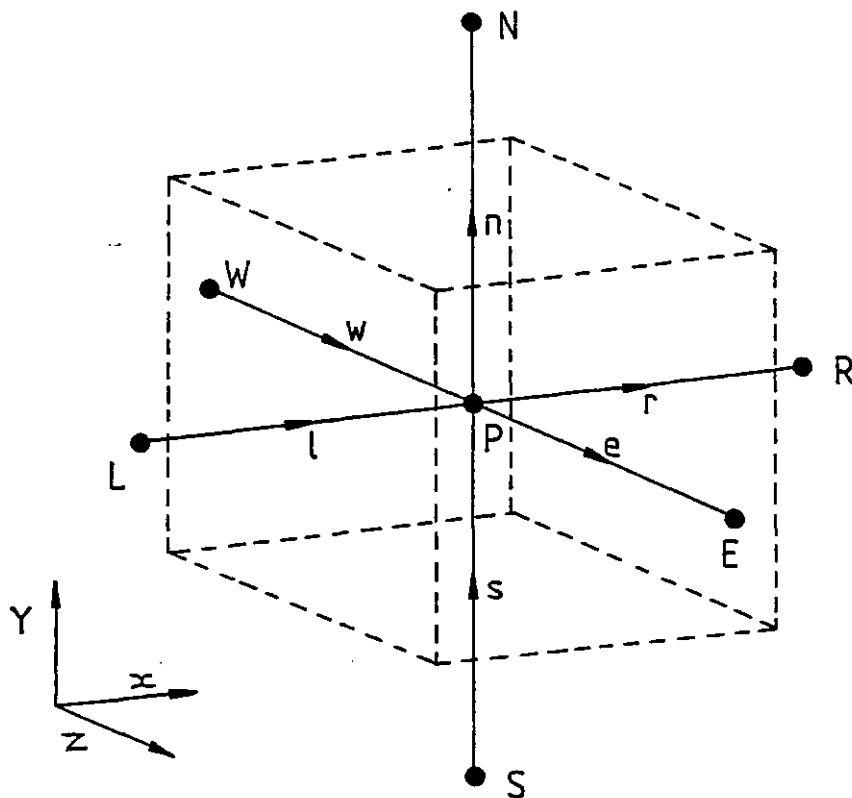


Fig. 4.1(a) Three-dimensional control volume

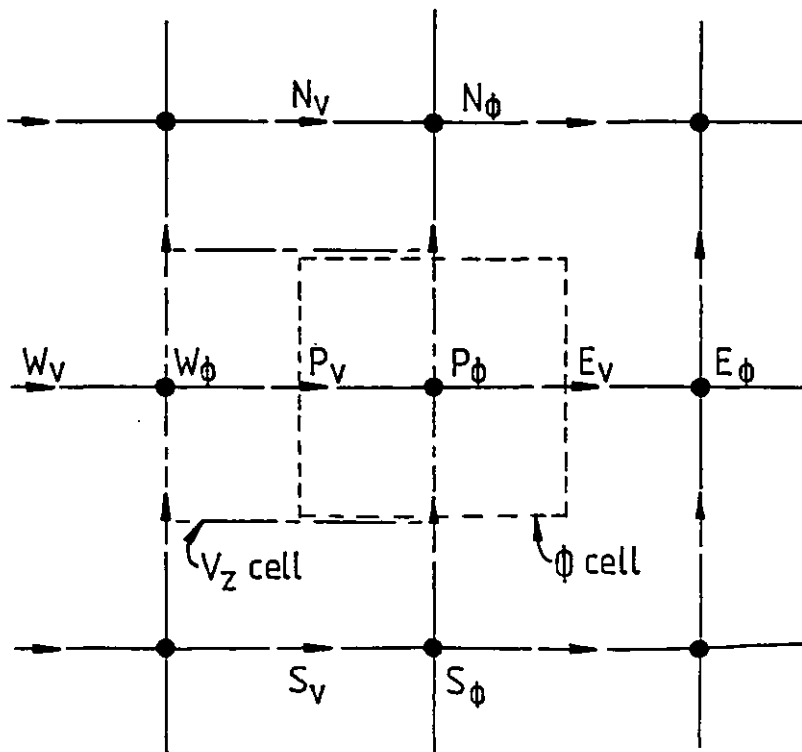


Fig. 4.1(b) Specification of ϕ and V_z cells

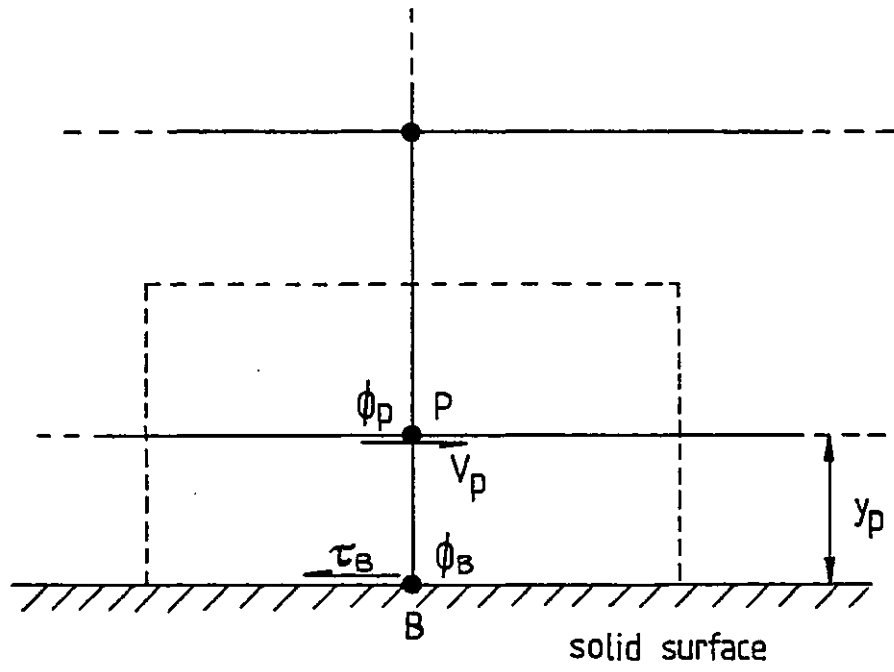


Fig. 4.2 Cell with solid surface boundary

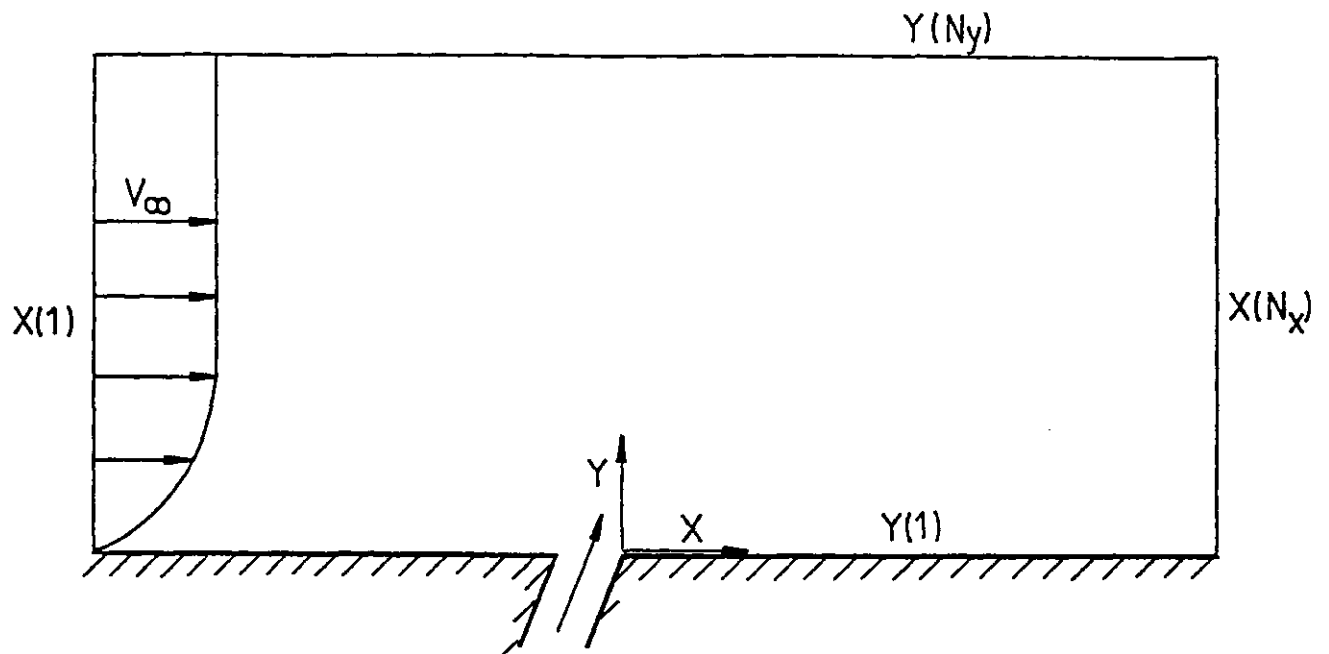


Fig. 5.1 Calculation domain for flush slot computations

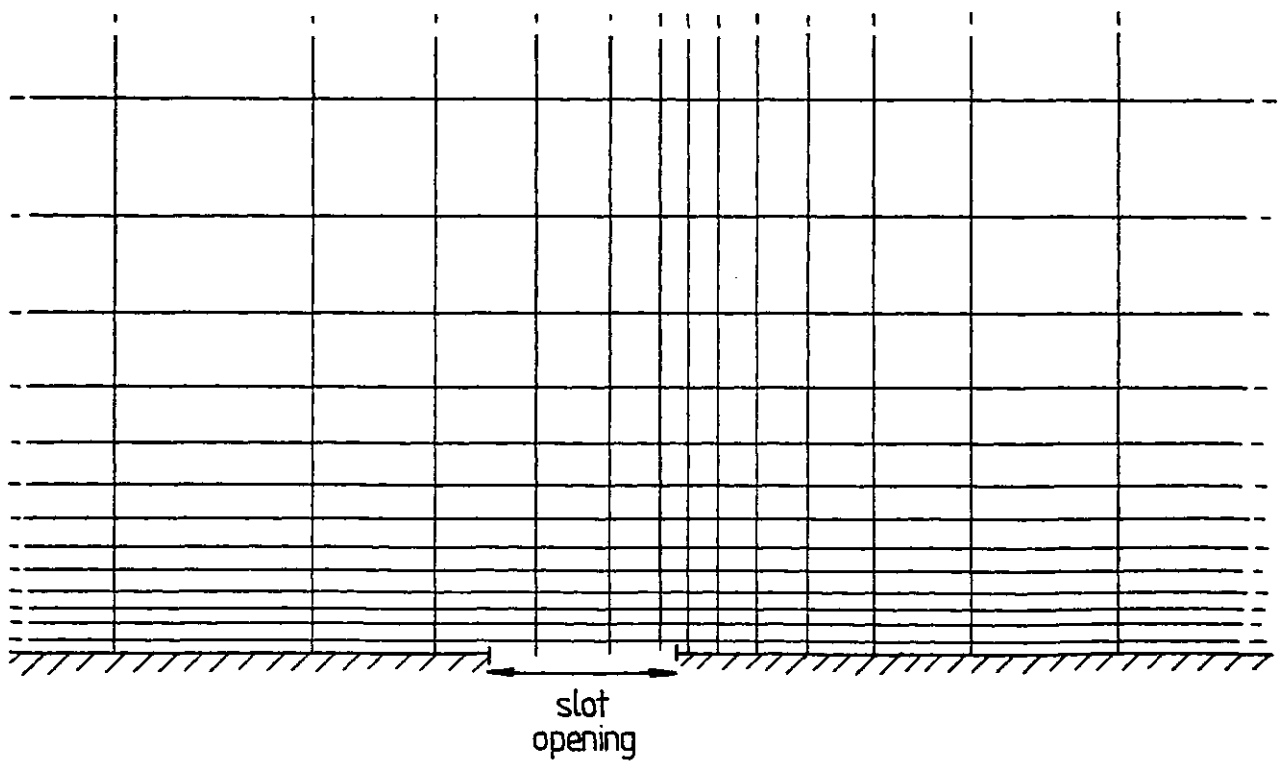


Fig. 5.2 Grid form employed for flush slot computations

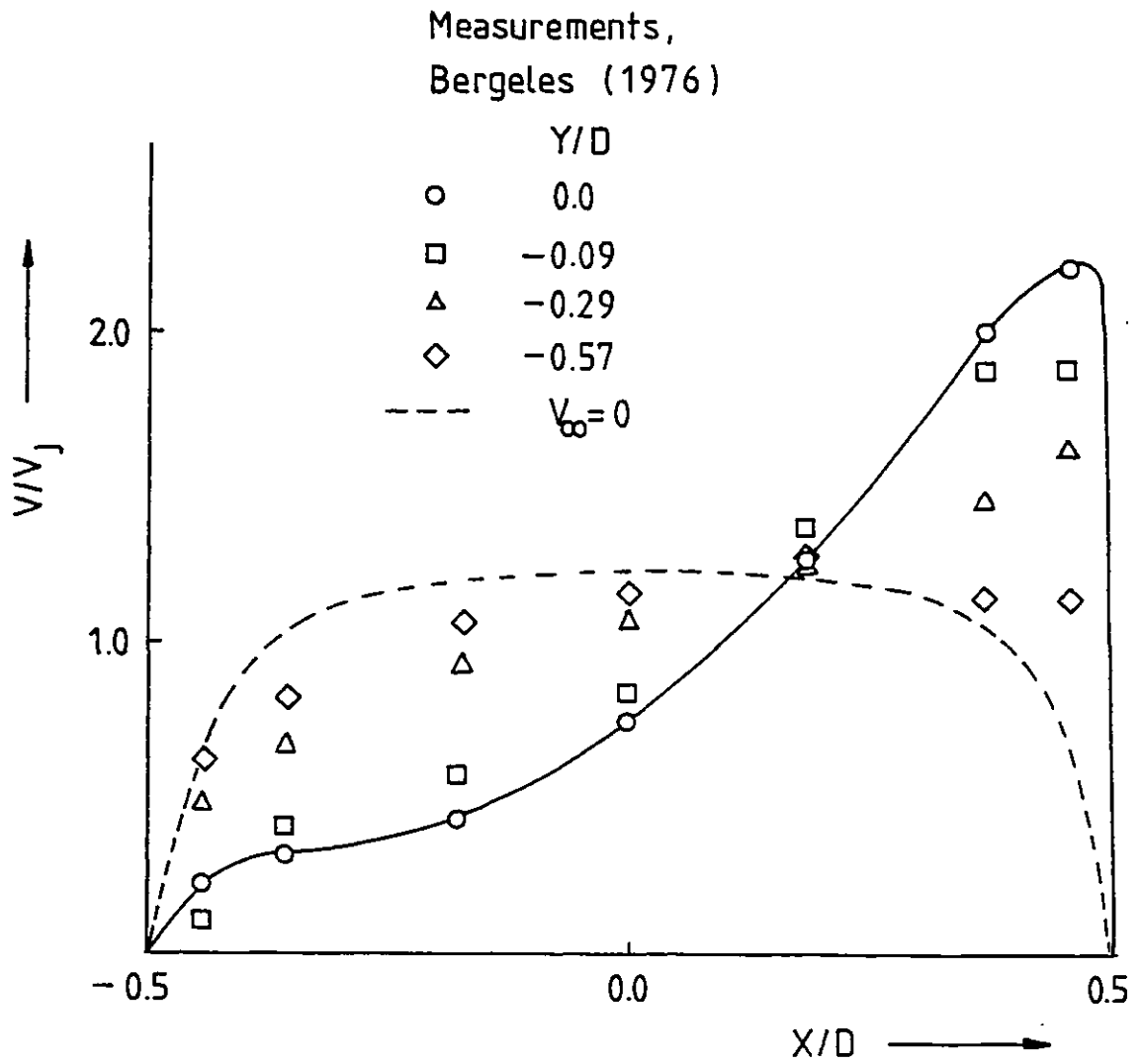


Fig. 5.3 Velocity profile in exit region of secondary jet, $J = 0.24$, $Z/D = 0.0$

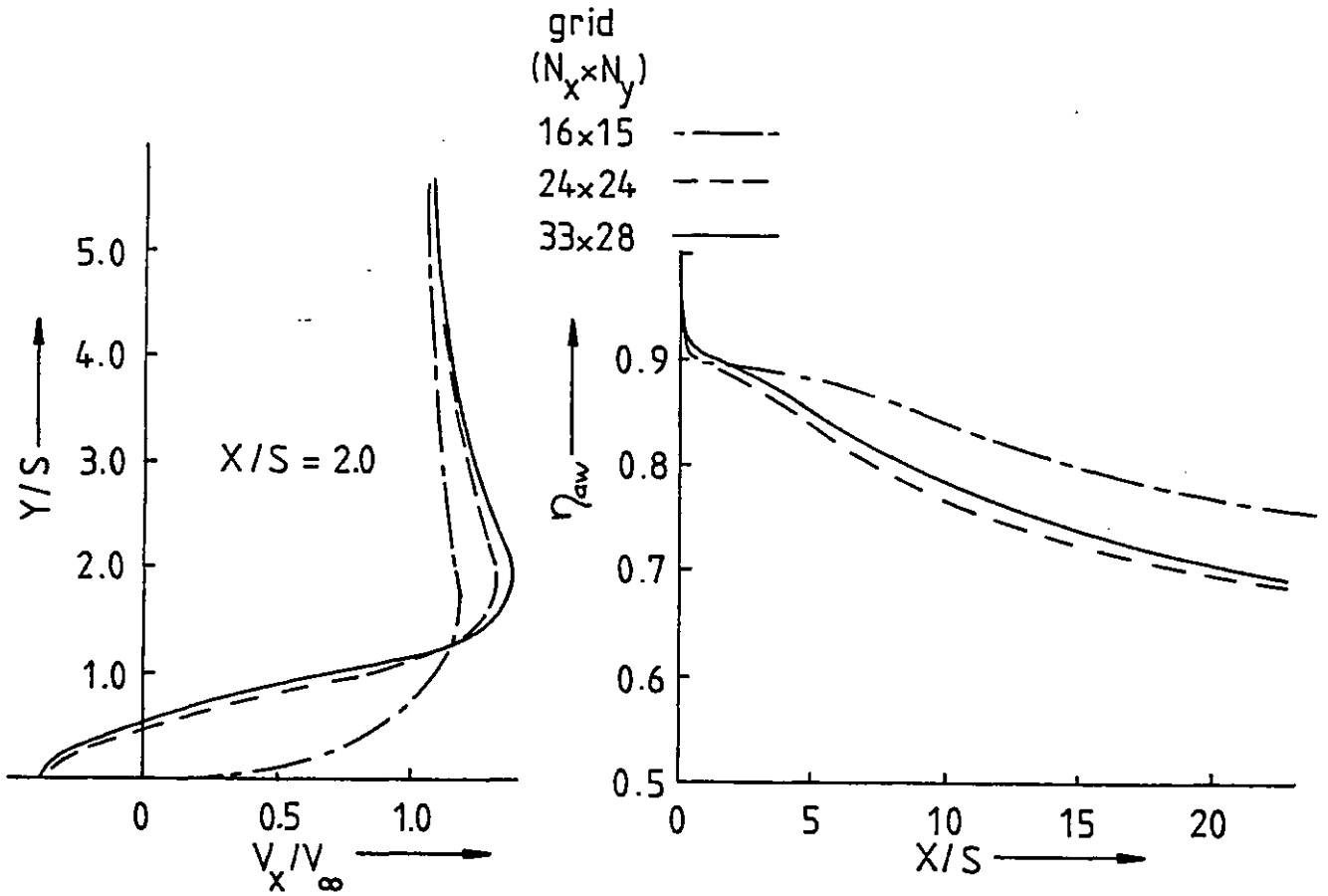


Fig. 5.4 Effect of grid refinement, $\alpha = 60^\circ$, $J = 1.45$

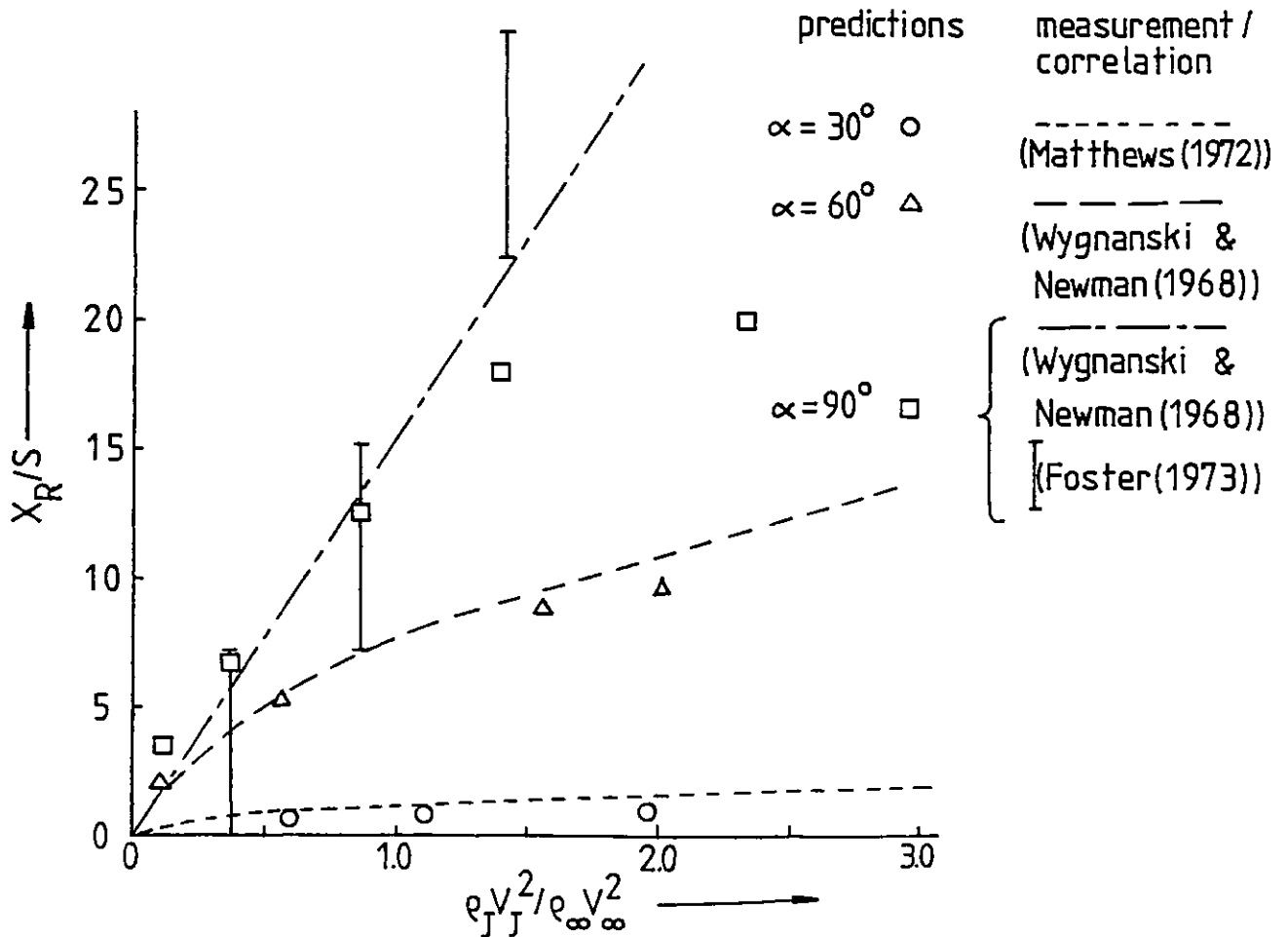


Fig. 5.5 Reattachment distance downstream of slot

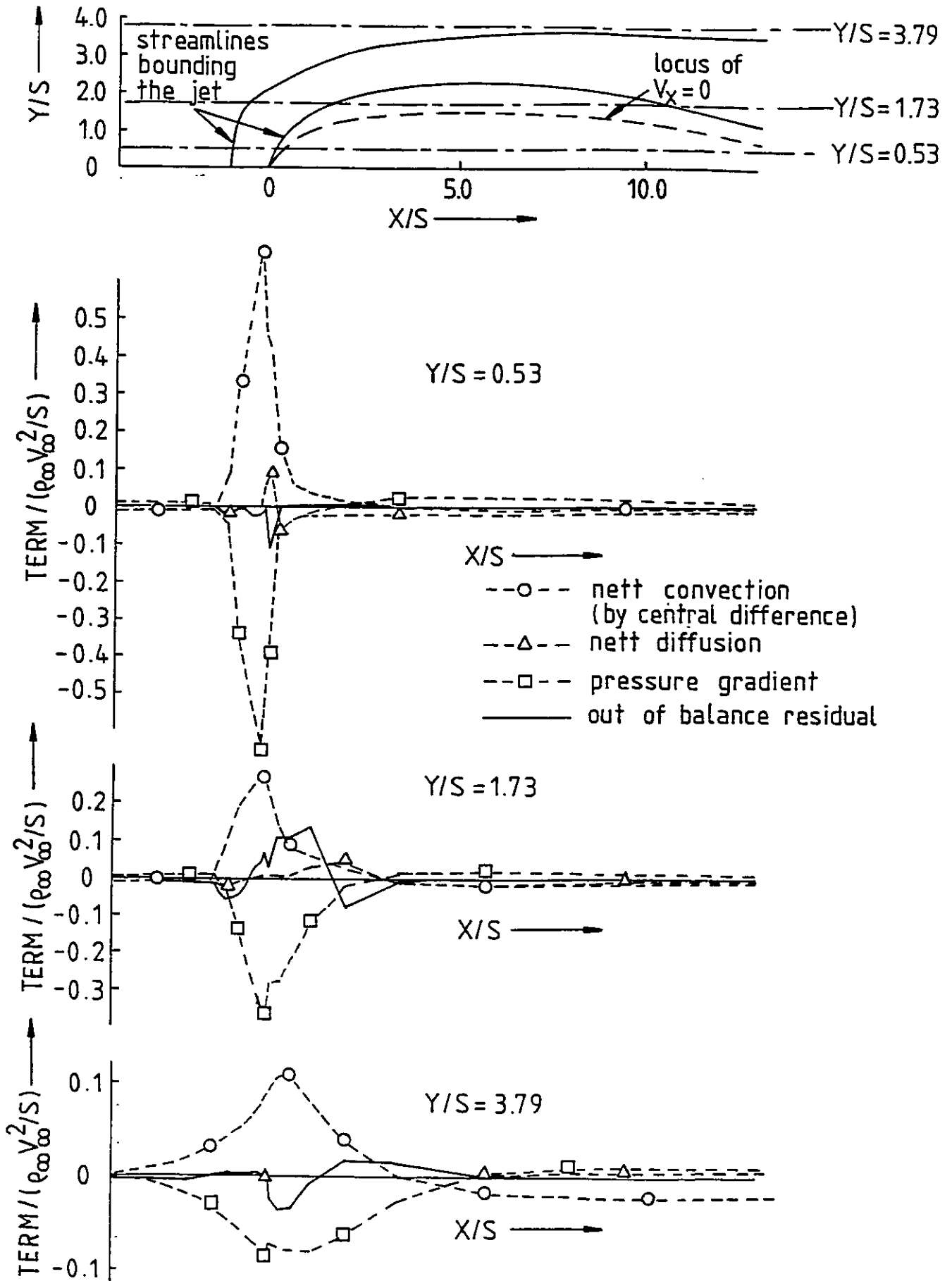


Fig. 5.6(a) Profiles of terms in the axial momentum equation,
 $\alpha = 90^\circ$, $J = 1.24$

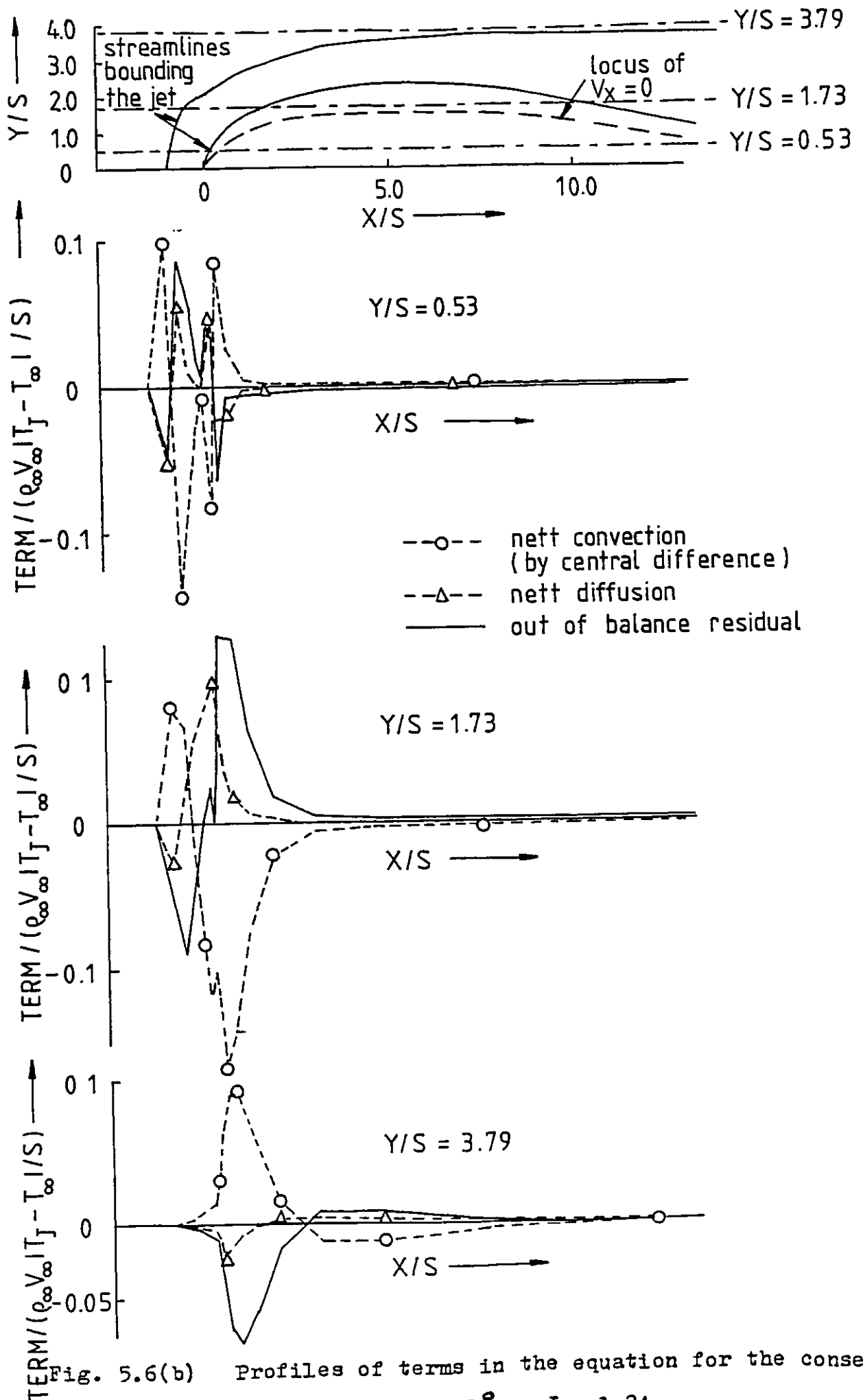


Fig. 5.6(b) Profiles of terms in the equation for the conserved scalar, T , $\alpha = 90^\circ$, $J = 1.24$

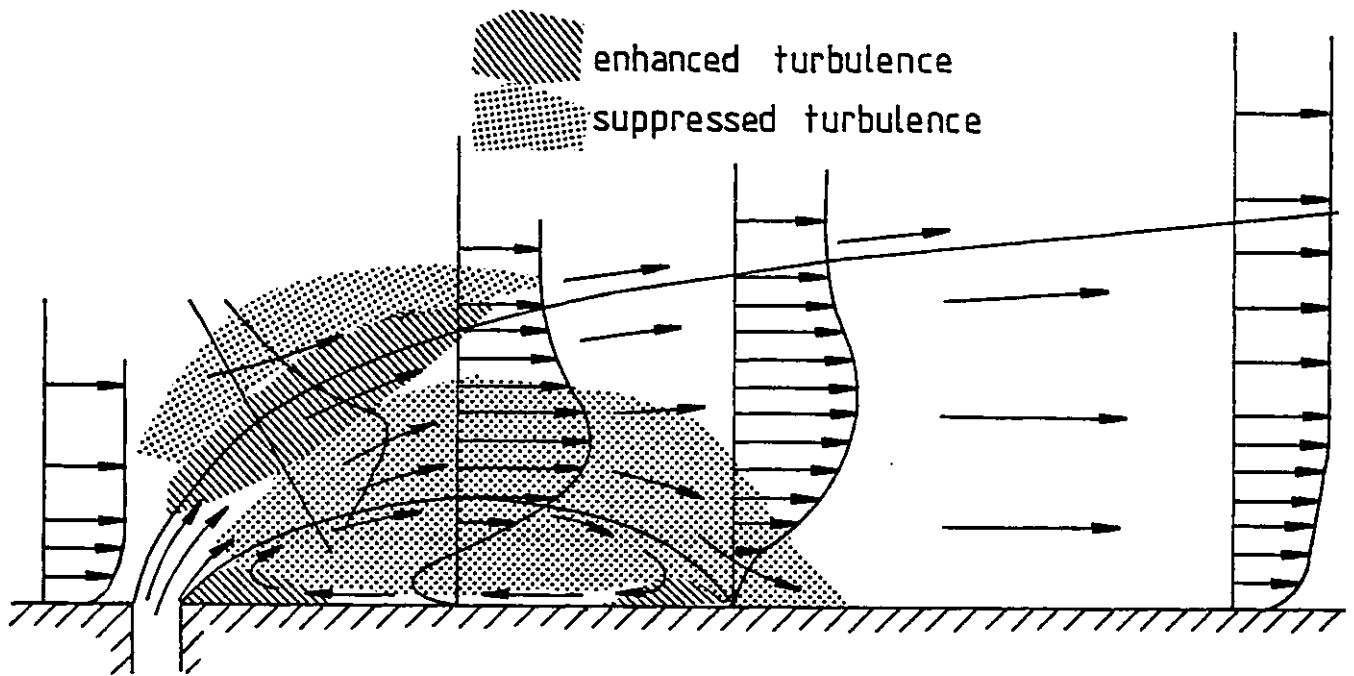


Fig. 5.7 Estimation of the regions of influence of streamline curvature on turbulent mixing

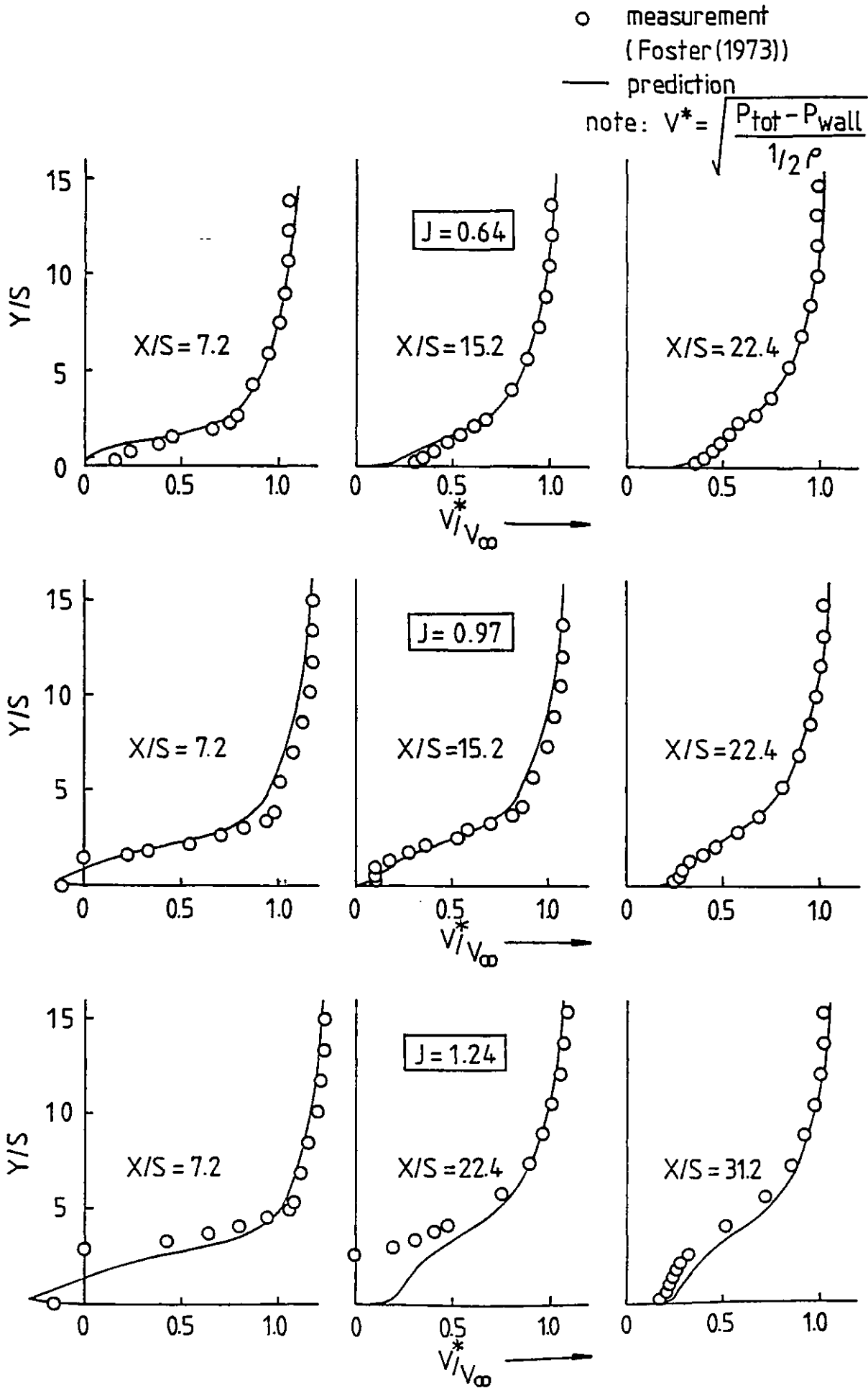
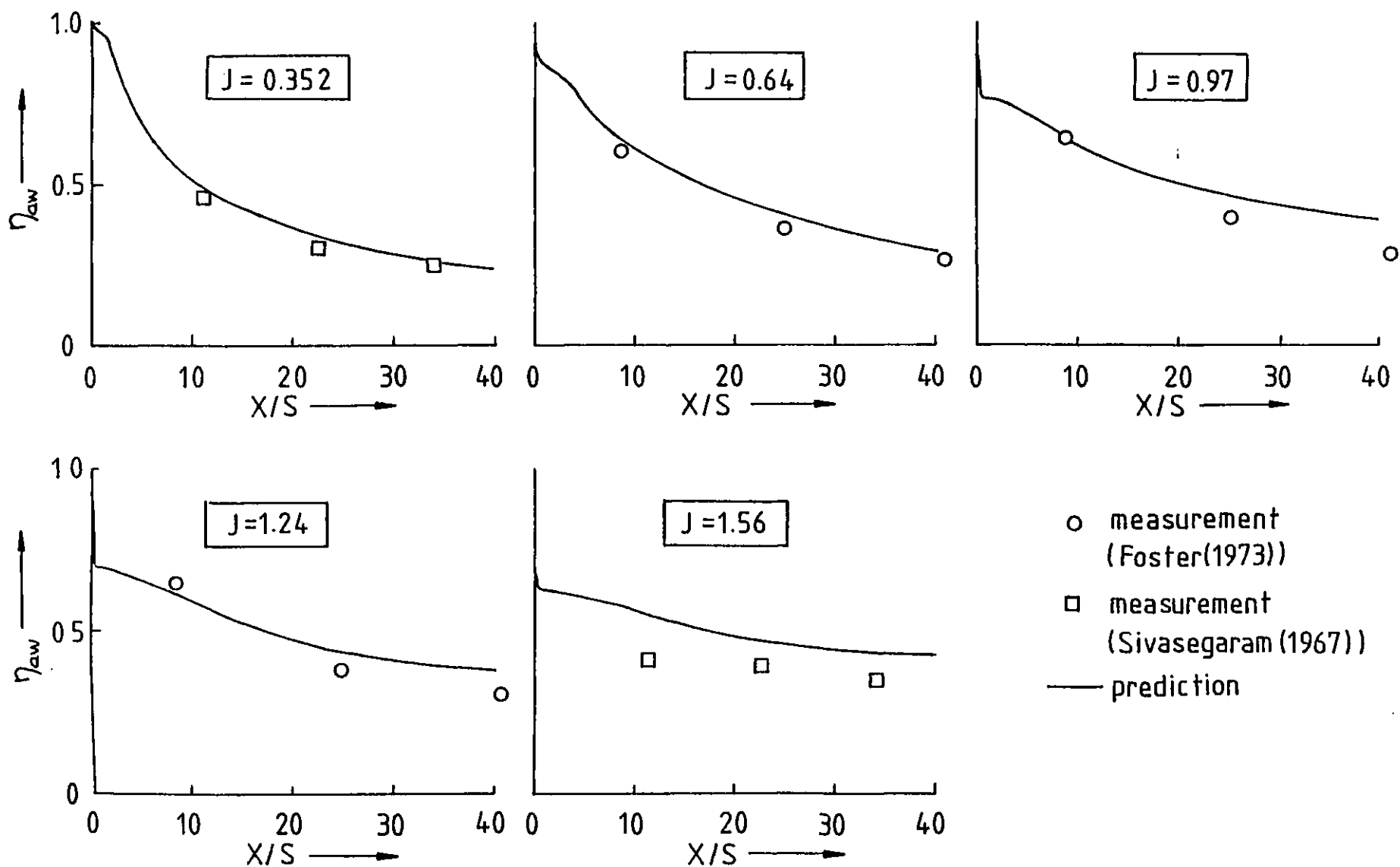


Fig. 5.8 Velocity profiles, $\alpha = 90^\circ$

Fig. 5.9 Adiabatic wall effectiveness, $\alpha = 90^\circ$



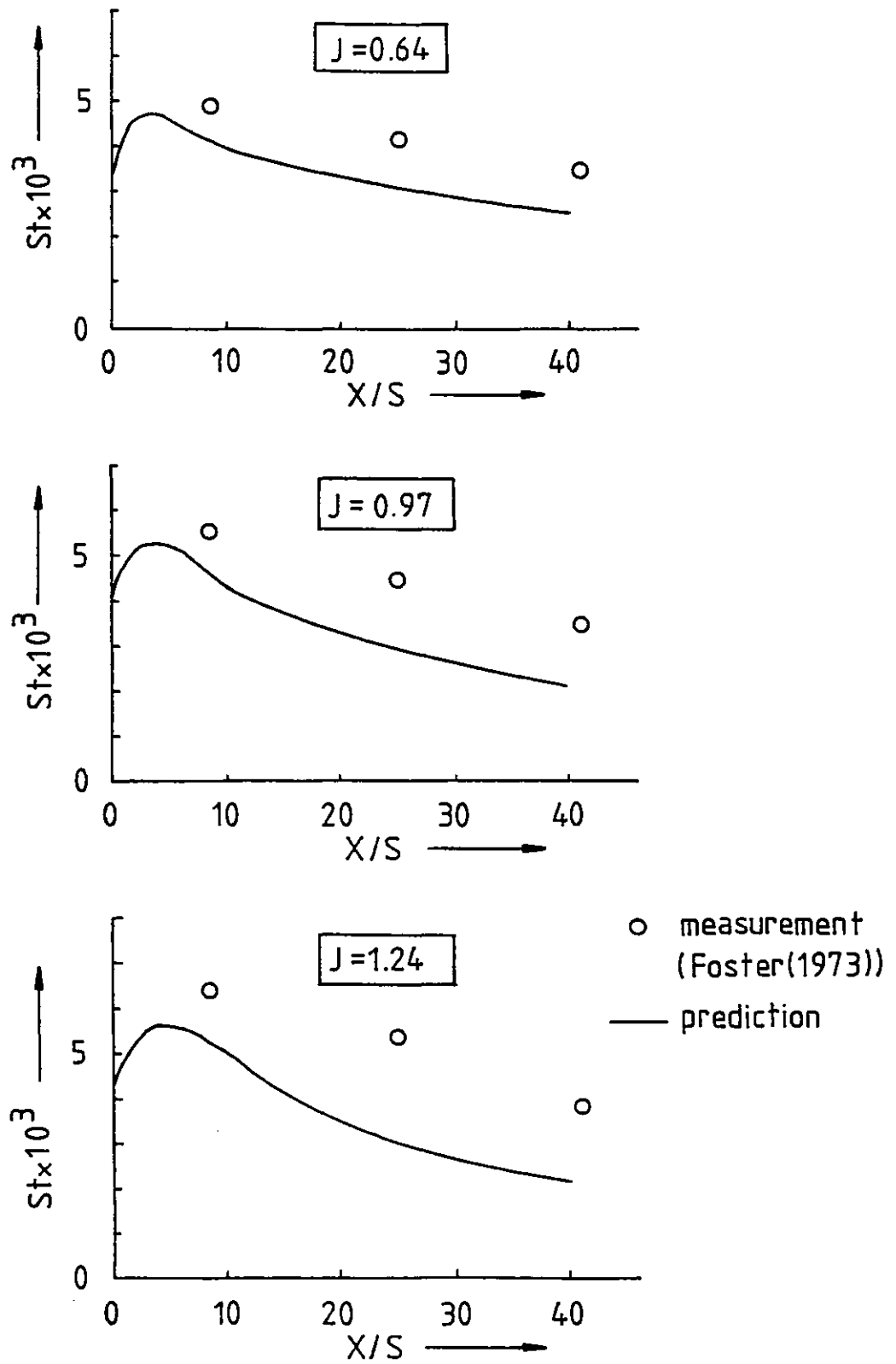


Fig. 5.10 Stanton number, $\alpha = 90^\circ$

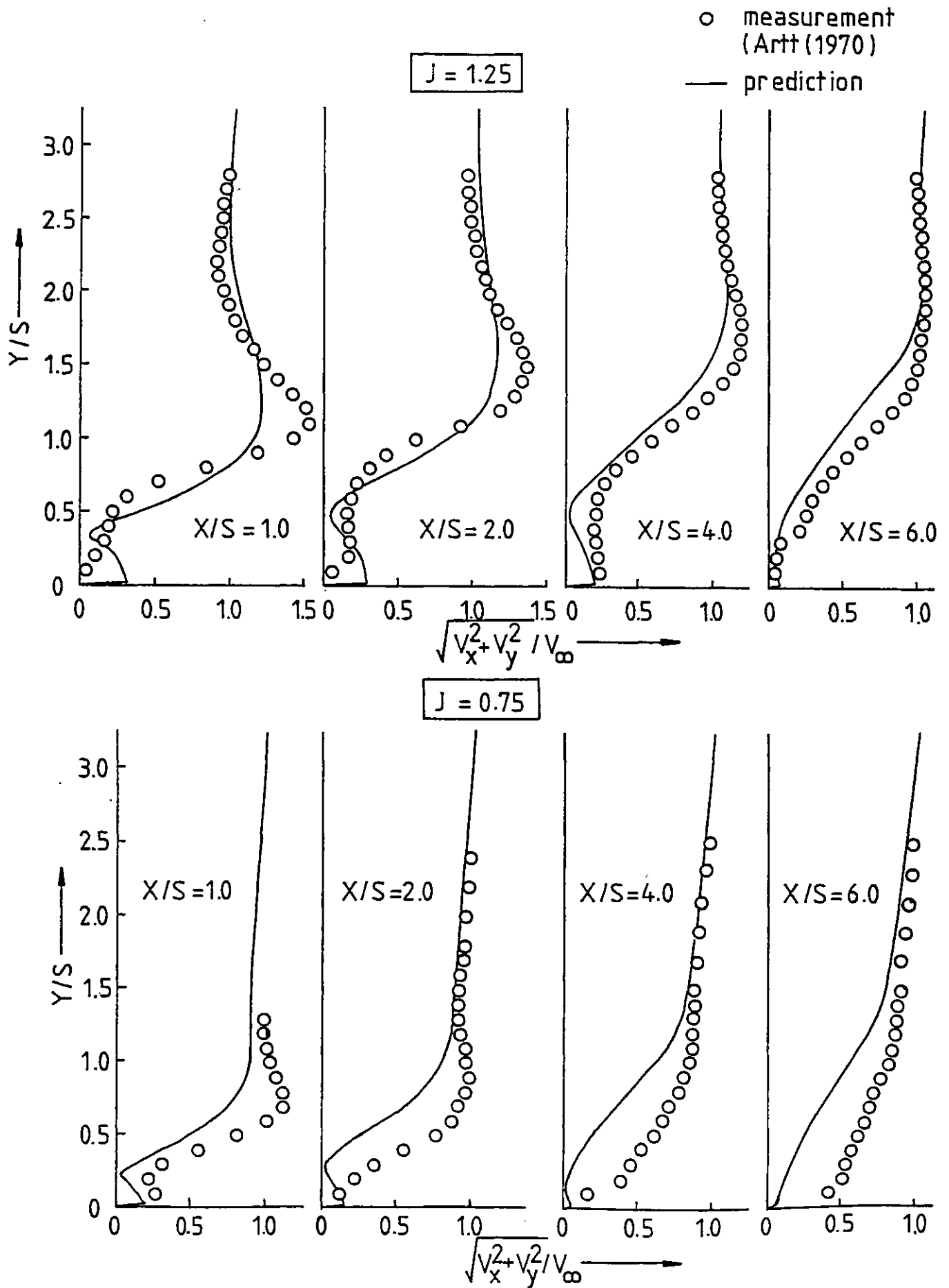


Fig. 5.11 Velocity profiles, $\alpha = 60^\circ$

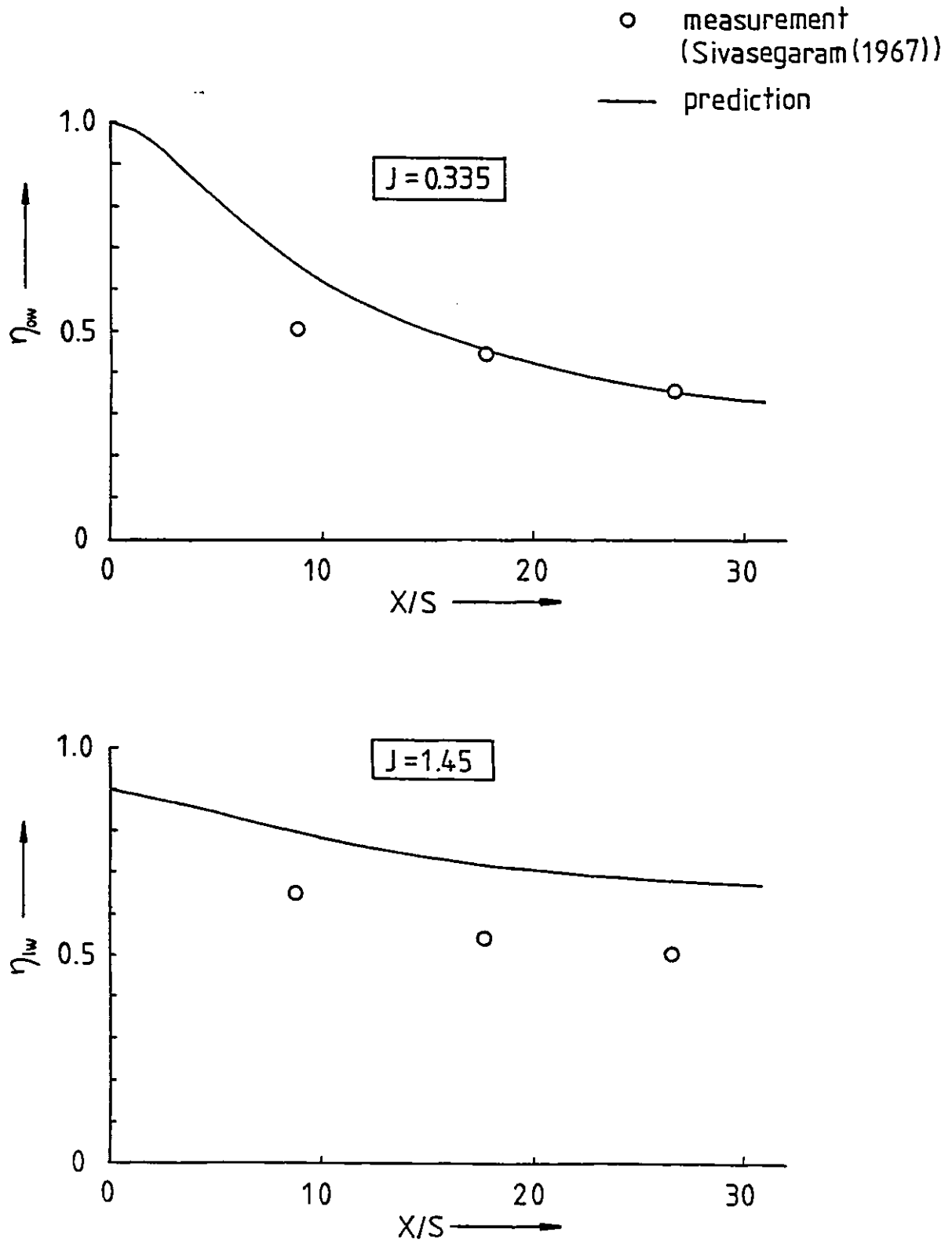


Fig. 5.12 Adiabatic wall effectiveness, $\alpha = 60^\circ$

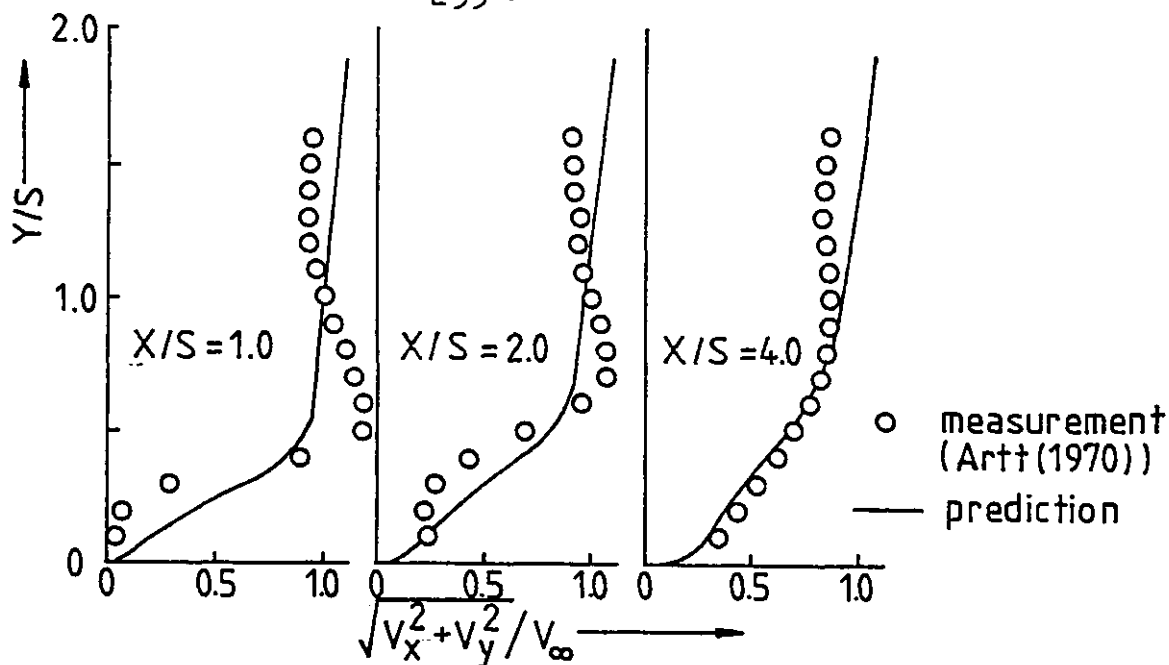


Fig. 5.13 Velocity profiles, $\alpha = 30^\circ$

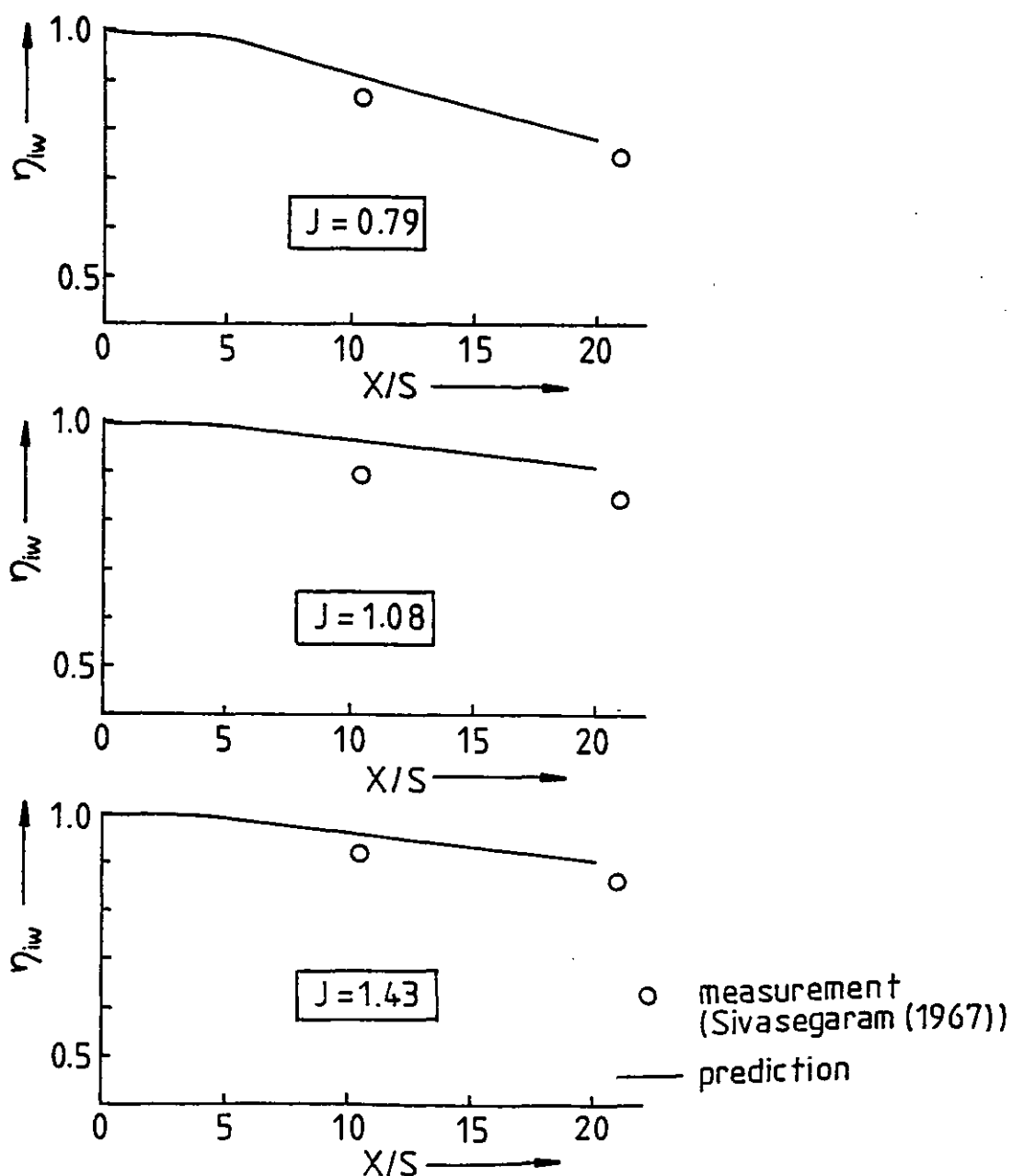


Fig. 5.14 Impervious wall effectiveness, $\alpha = 30^\circ$

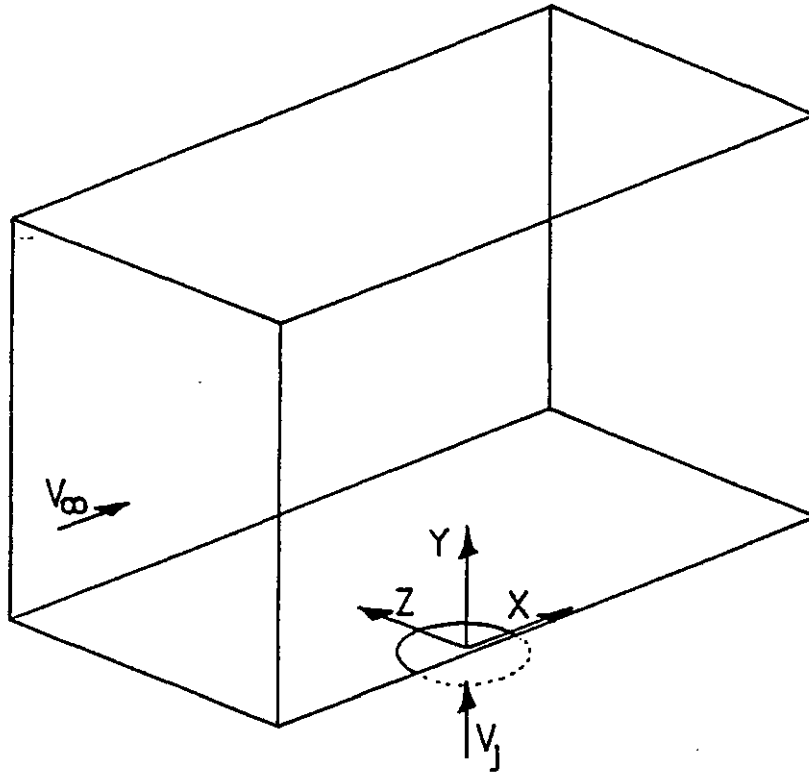


Fig. 6.1 Calculation domain for the jet-mainstream interaction

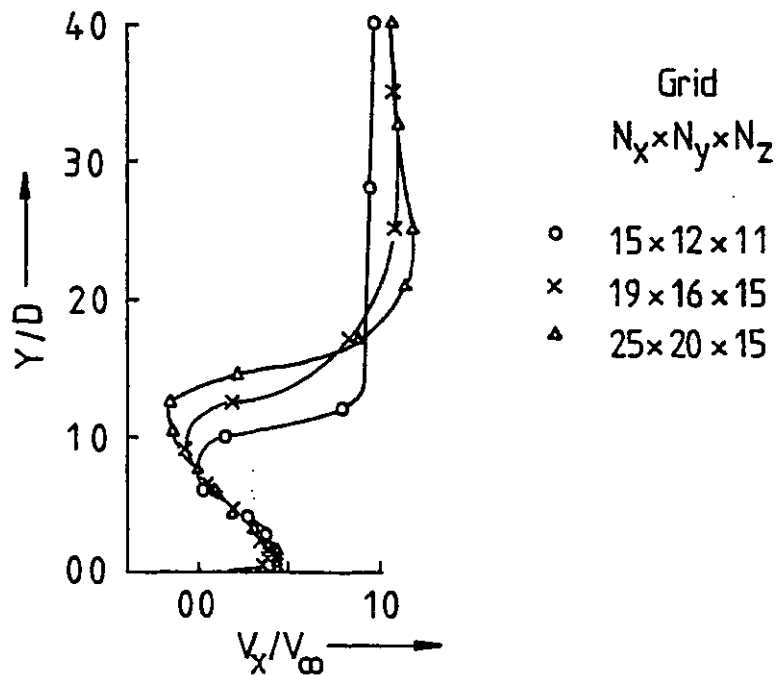


Fig. 6.2 Effect of grid refinement on the V_x - velocity profile, $J = 2.0$, $x/D = 1.4$, $z/D = 0.0$

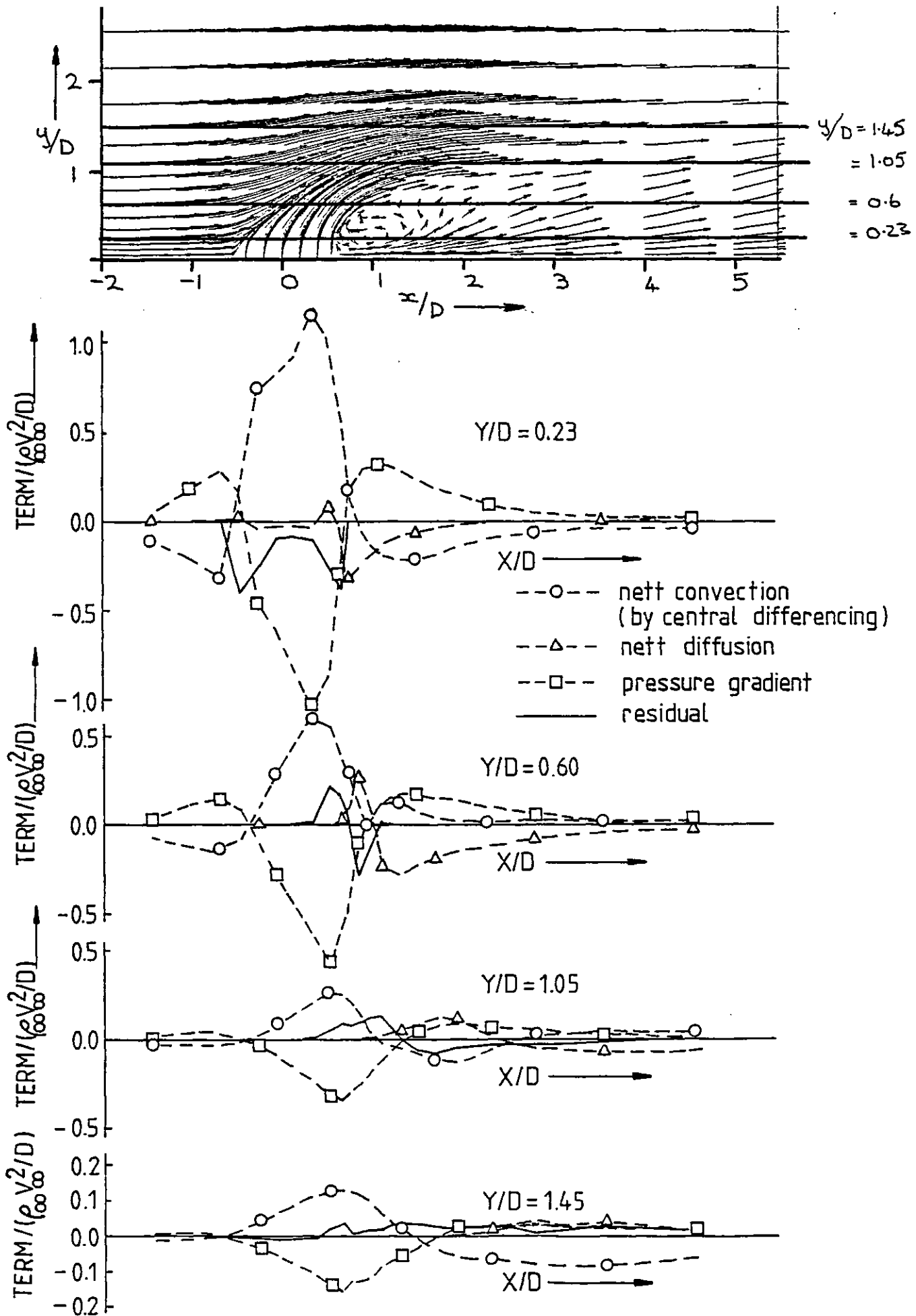


Fig. 6.3(a) Profiles of terms in the axial momentum equation,
 $J = 1.0$, $Z/D = 0.25$

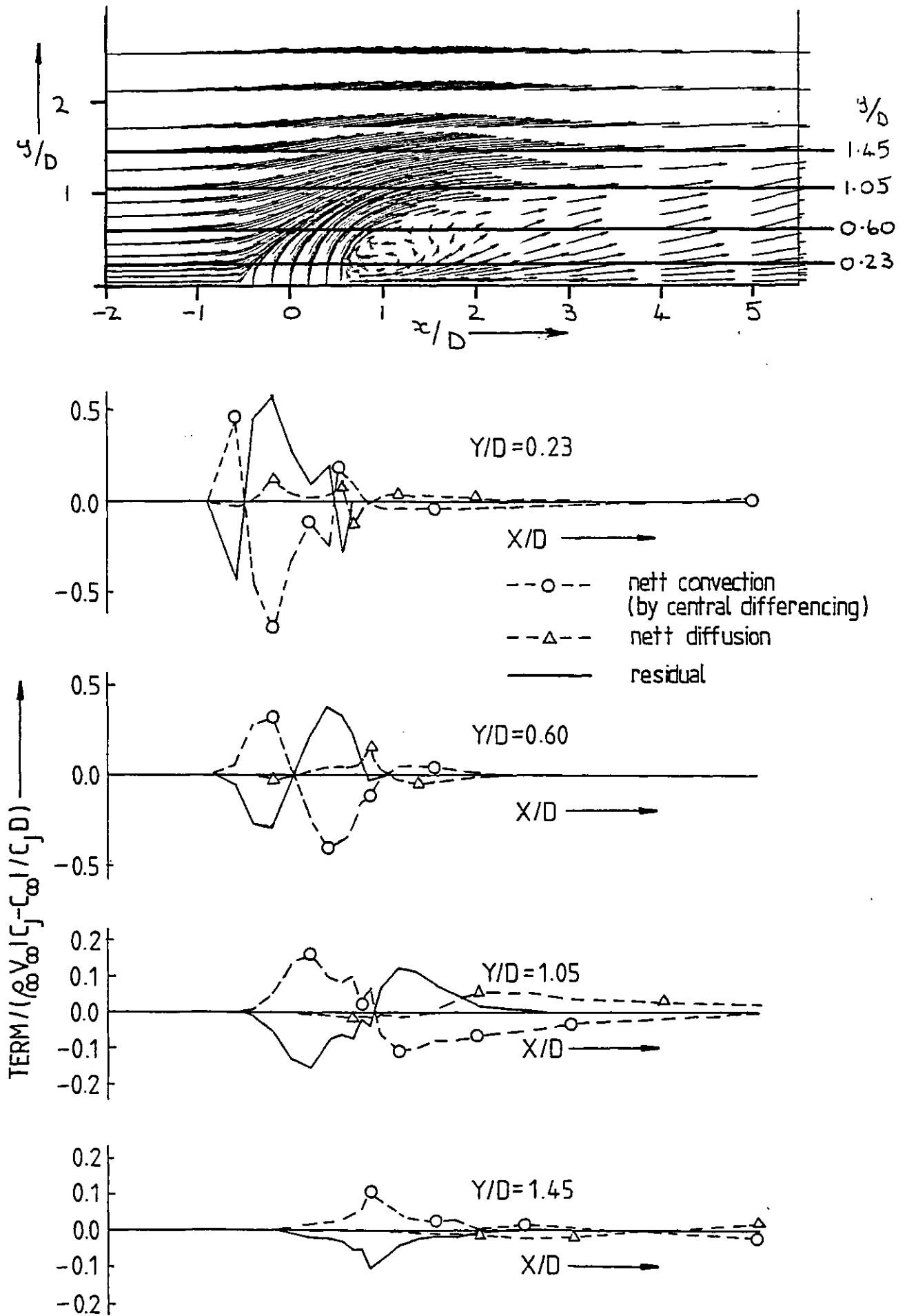


Fig. 6.3(b) Profiles of terms in the equation for the conserved scalar, C , $J = 1.0$, $Z/D = 0.25$

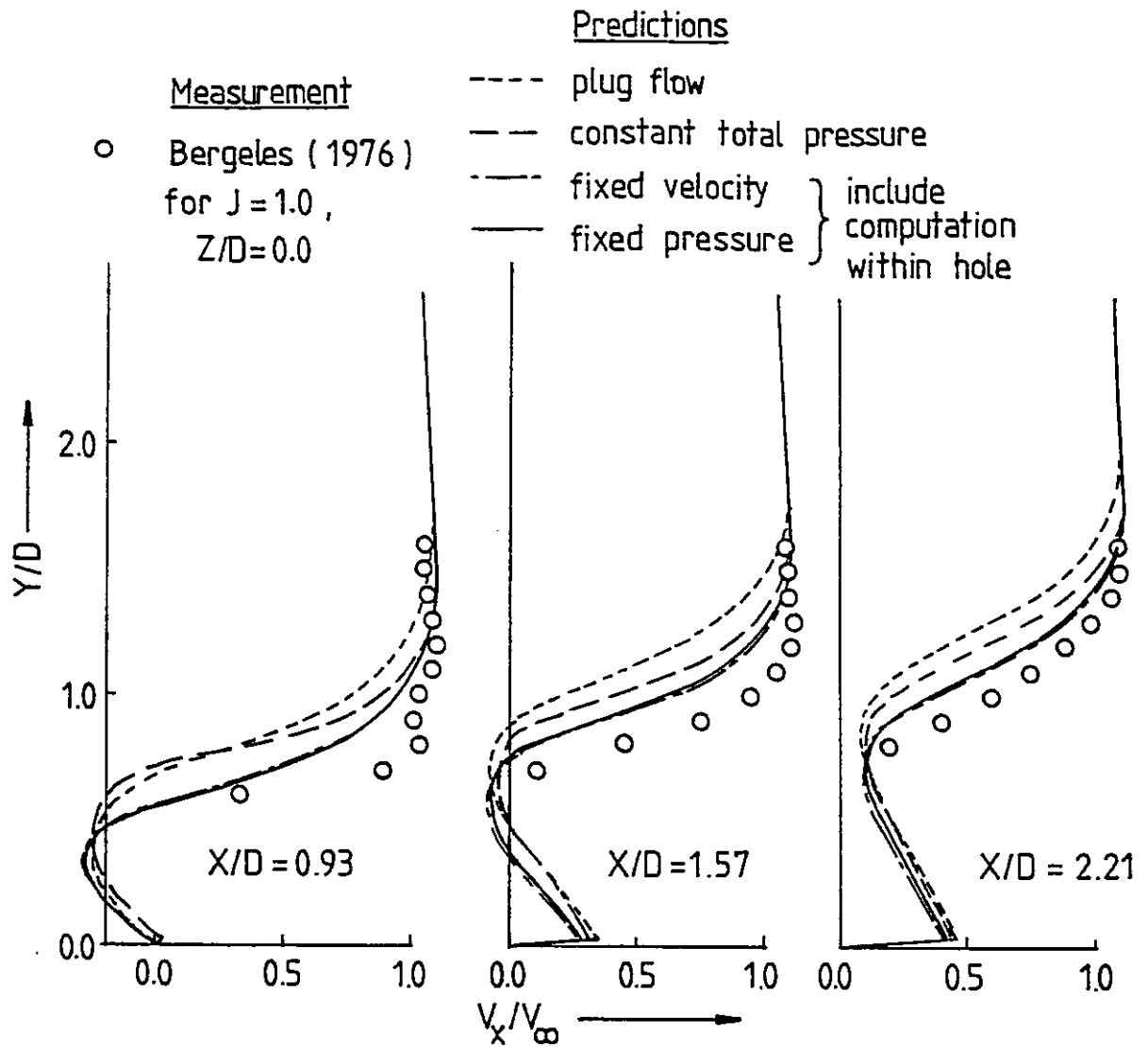


Fig. 6.4 Influence of hole exit boundary condition on V_x - velocity profiles

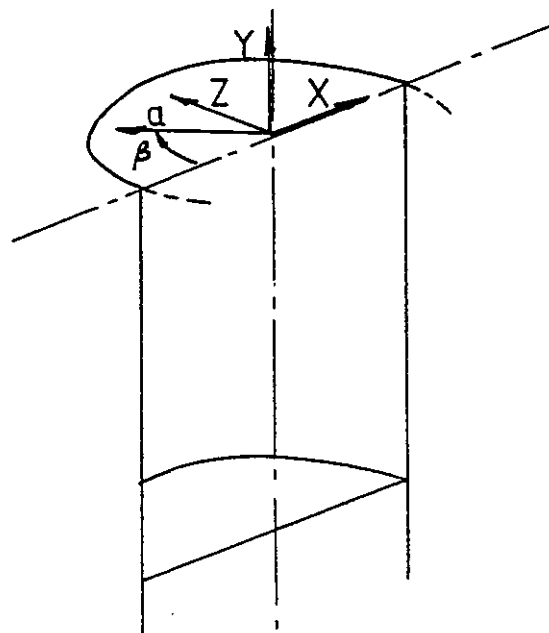


Fig. 6.5 Domain and co-ordinate system for computation within the hole

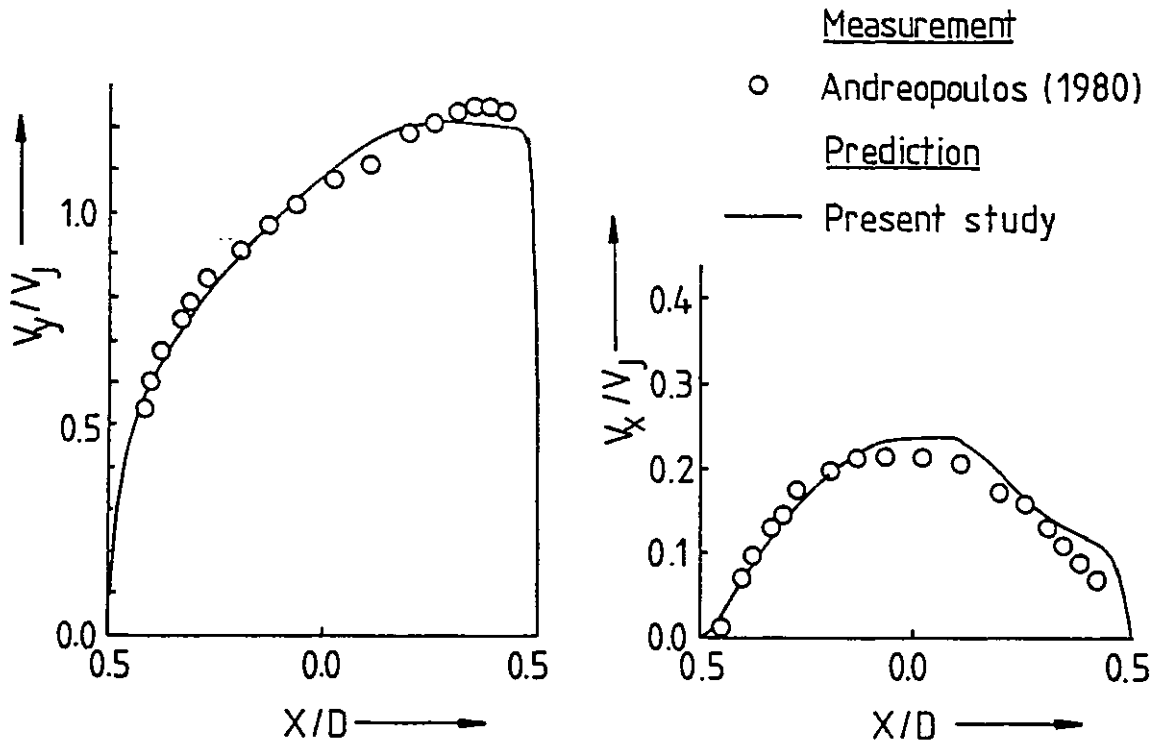


Fig. 6.6 Velocity profiles in hole exit plane, $J = 1.0$, $Z/D = 0.0$

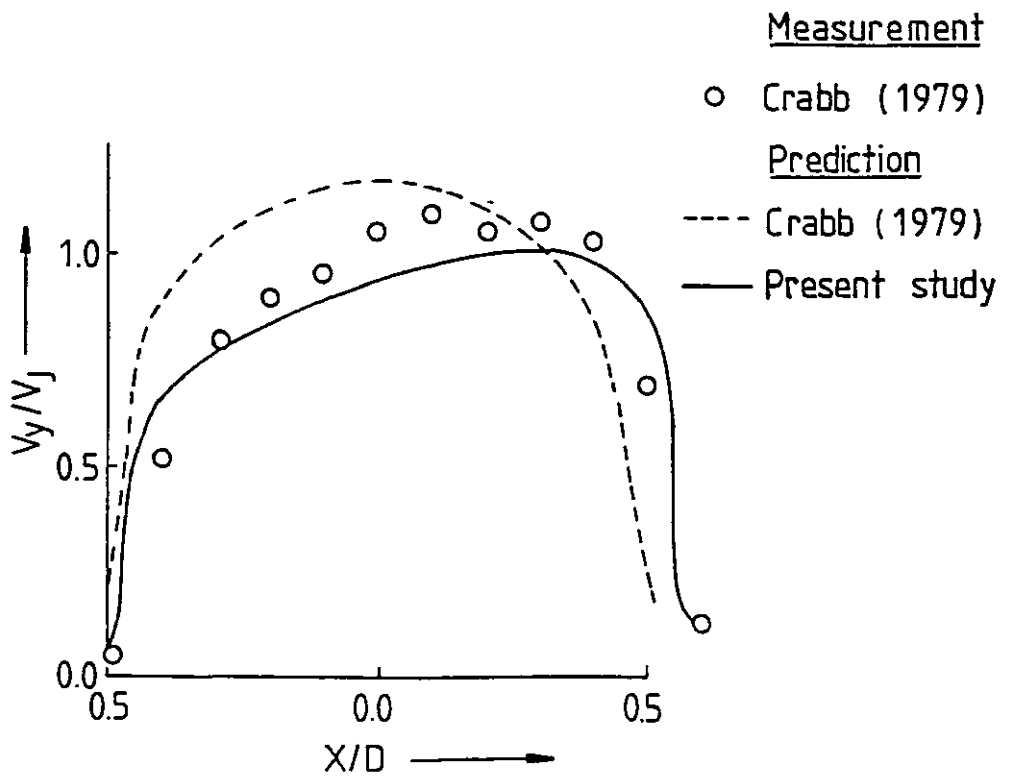


Fig. 6.7 V_y - velocity profile through jet, $J = 2.3$, $y/D = 0.25$, $Z/D = 0.0$

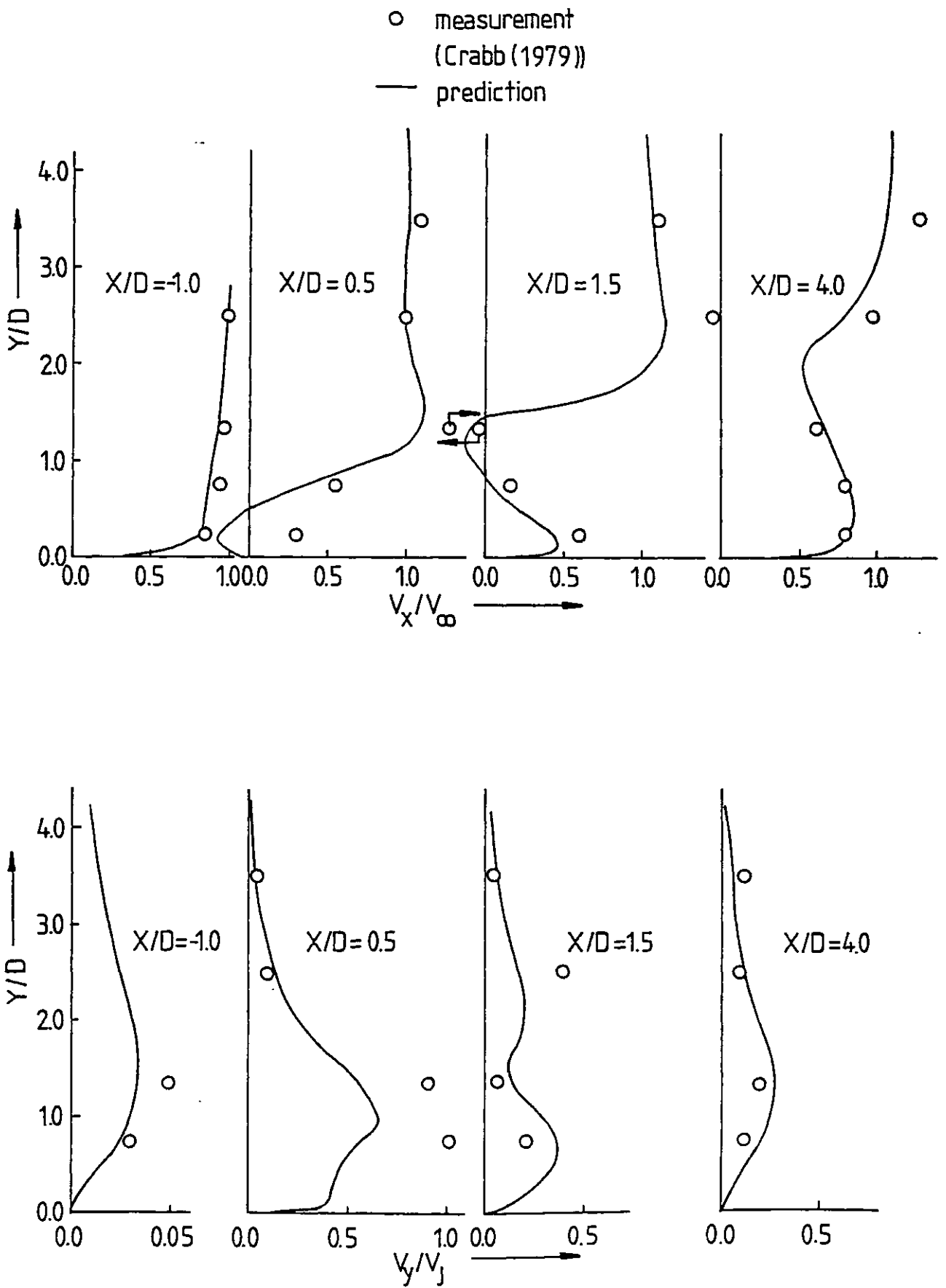


Fig. 6.8 Velocity profiles, $J = 2.3$, $Z/D = 0.0$

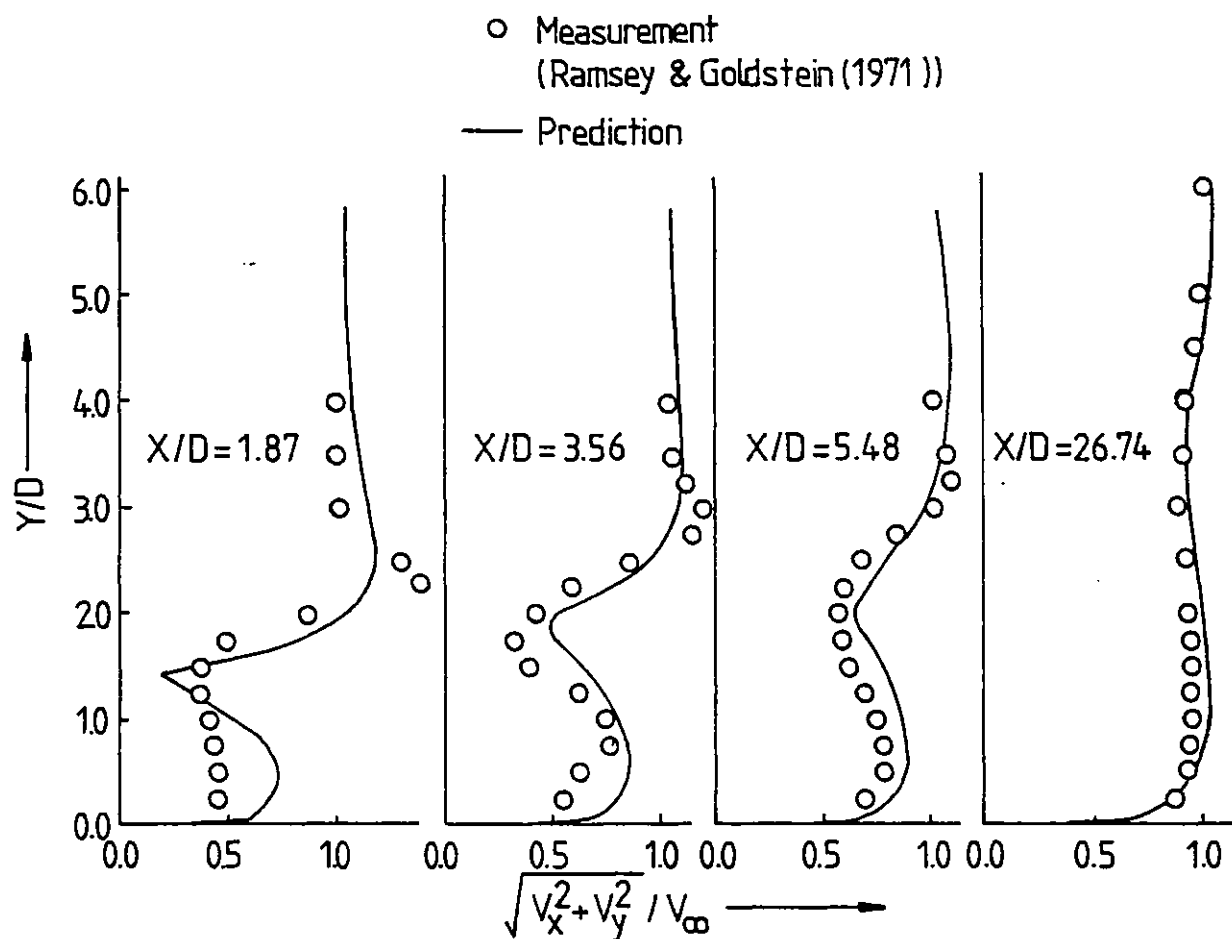


Fig. 6.9(a) Velocity profiles, $J = 2.0$, $Z/D = 0.0$

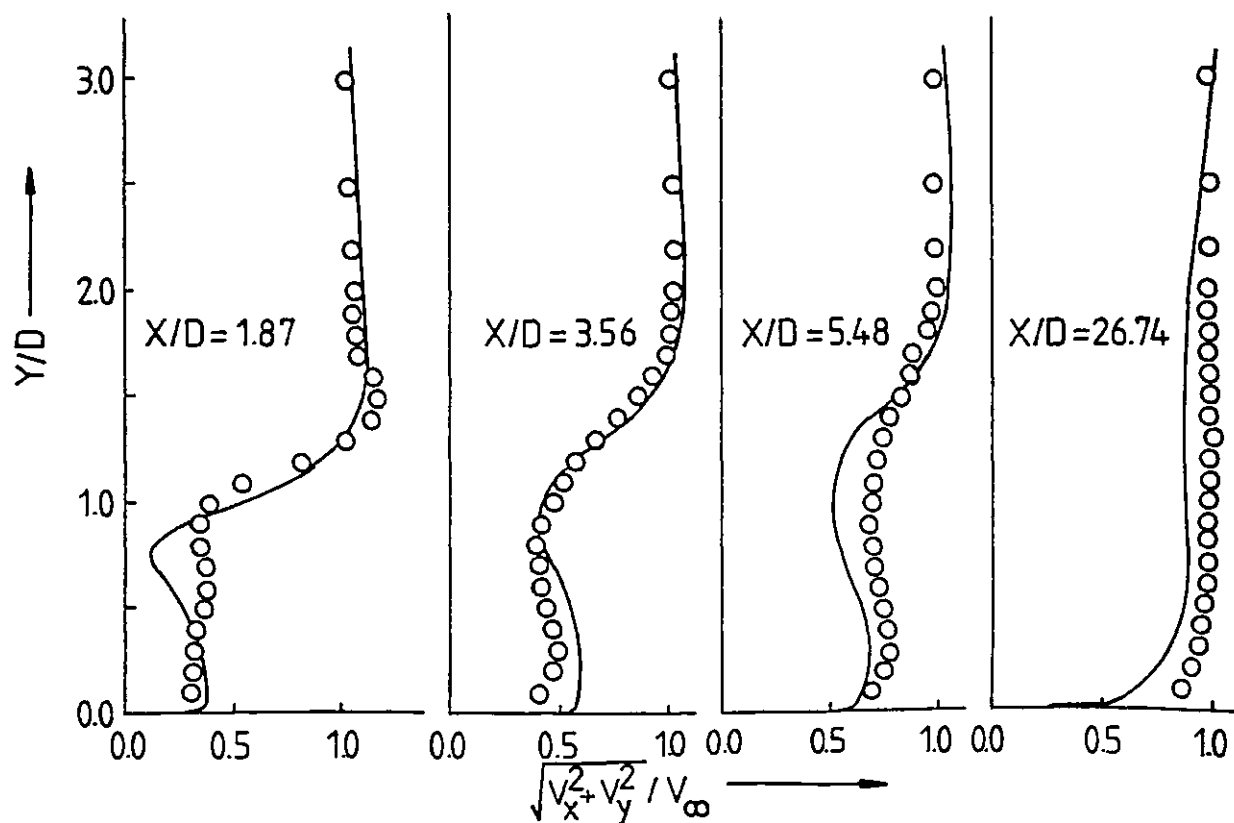


Fig. 6.9(b) Velocity profiles, $J = 1.0$, $Z/D = 0.0$

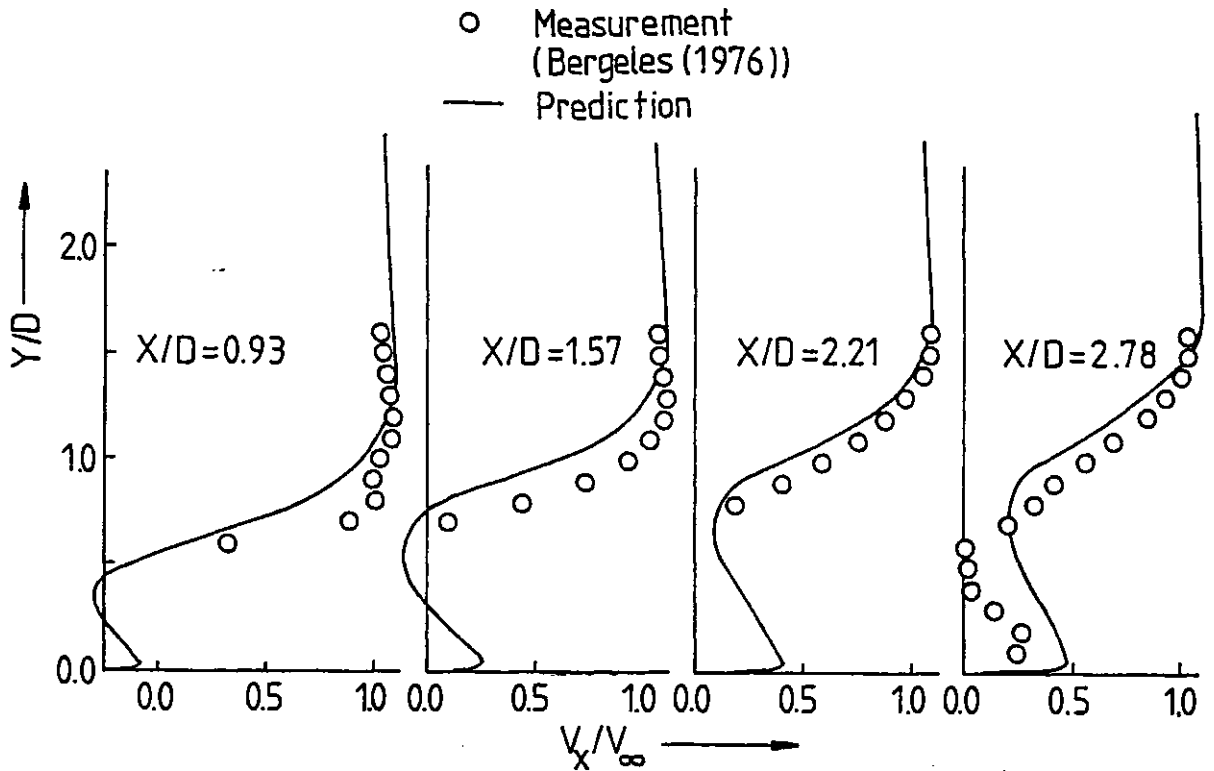


Fig. 6.10(a) Velocity profiles, $J = 1.0$, $Z/D = 0.0$

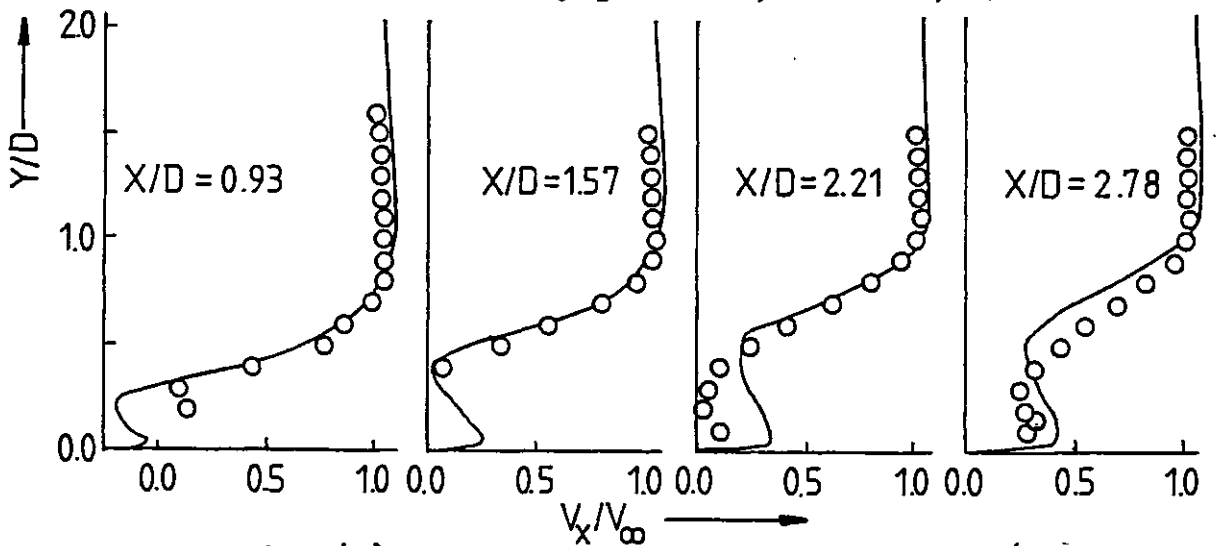


Fig. 6.10(b) Velocity profiles, $J = 0.5$, $Z/D = 0.0$

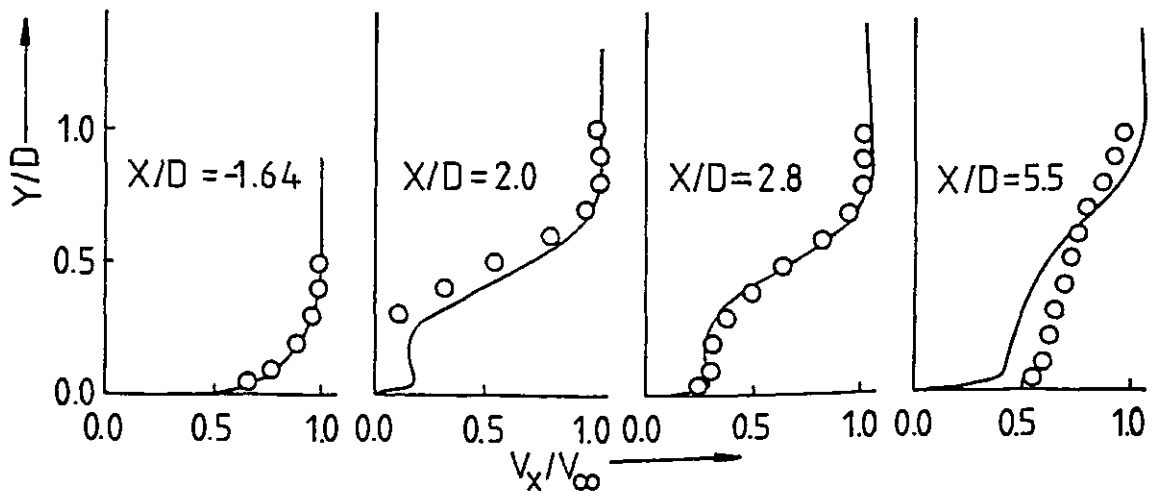
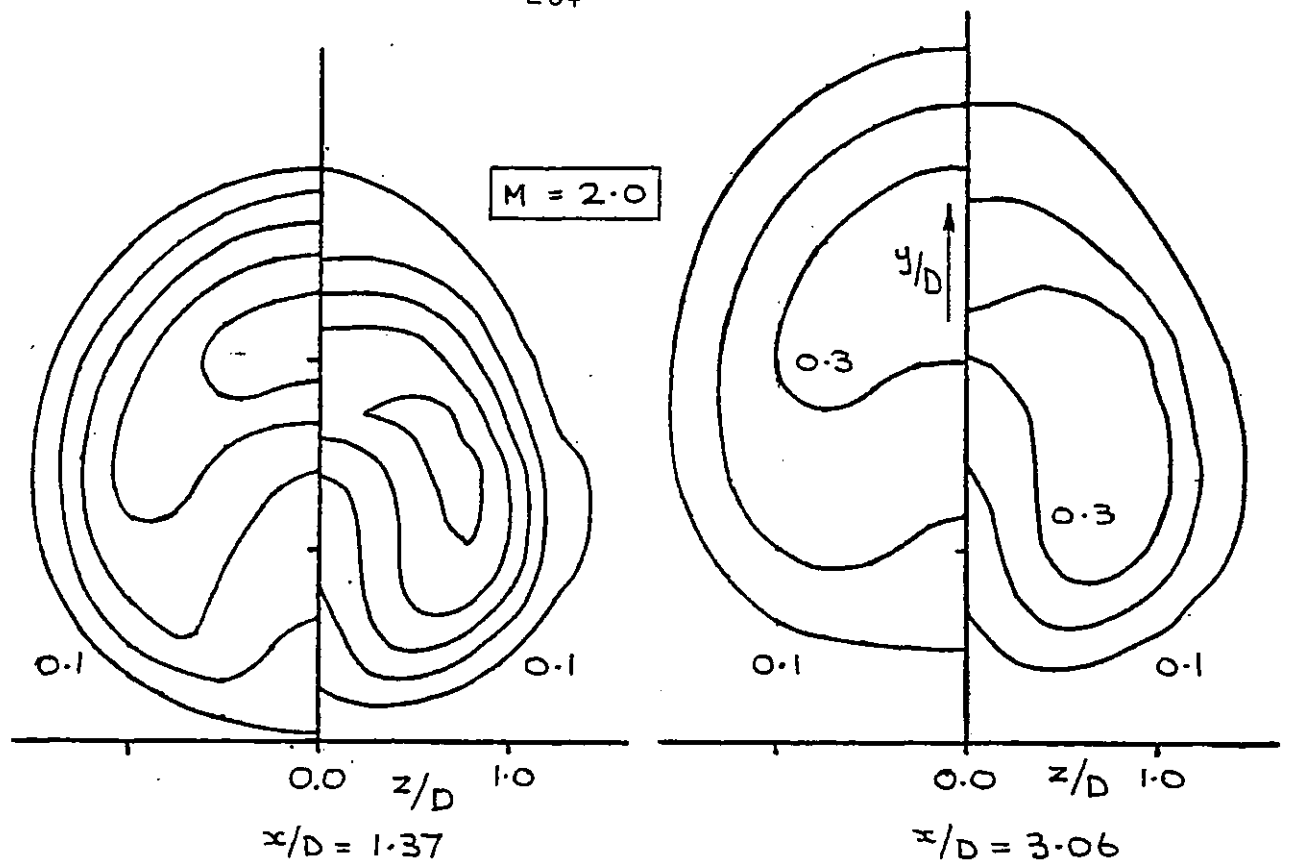
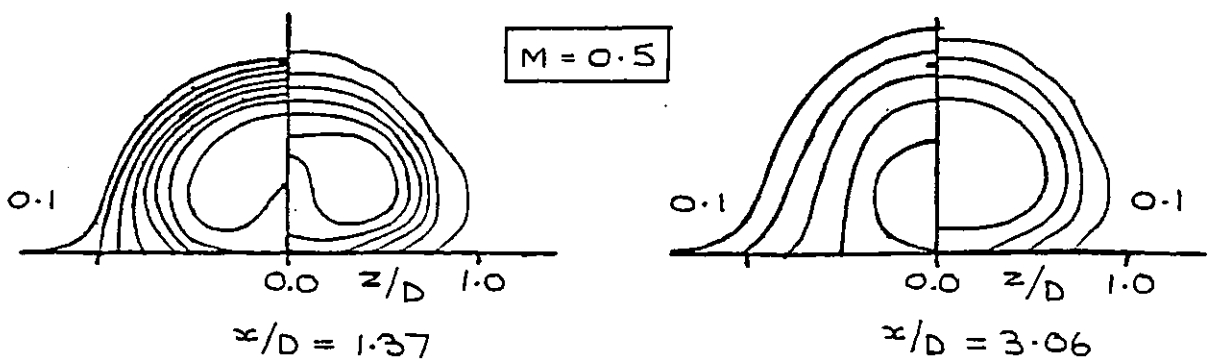
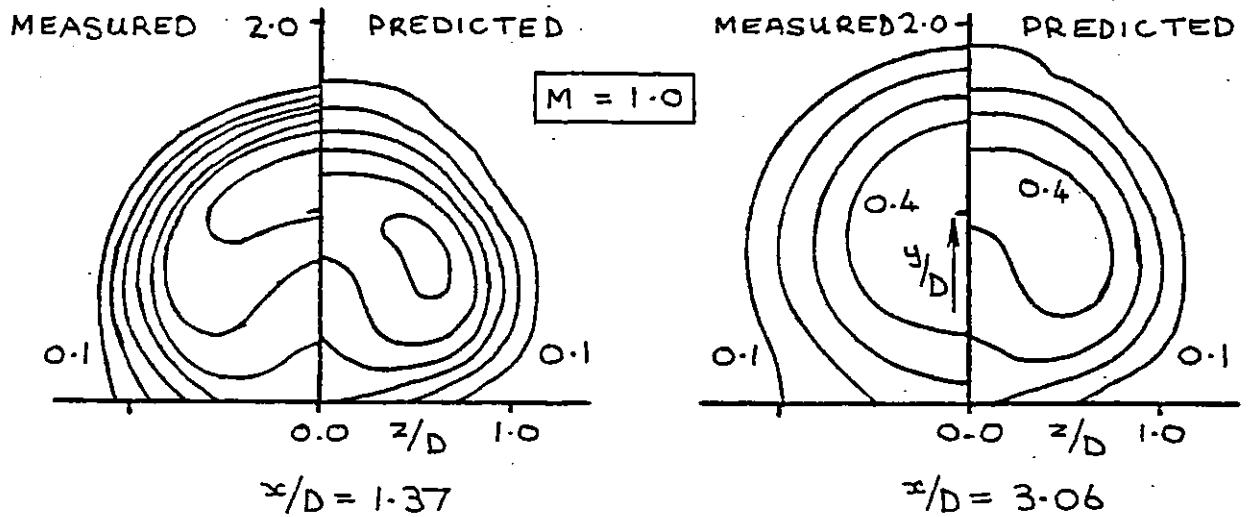


Fig. 6.10(c) Velocity profiles, $J = 0.24$, $Z/D = 0.0$



CONTOUR INTERVAL = 0.1



MEASUREMENTS ; RAMSEY AND GOLDSTEIN (1971)

Fig. 6.11 Effectiveness contours on cross-stream planes

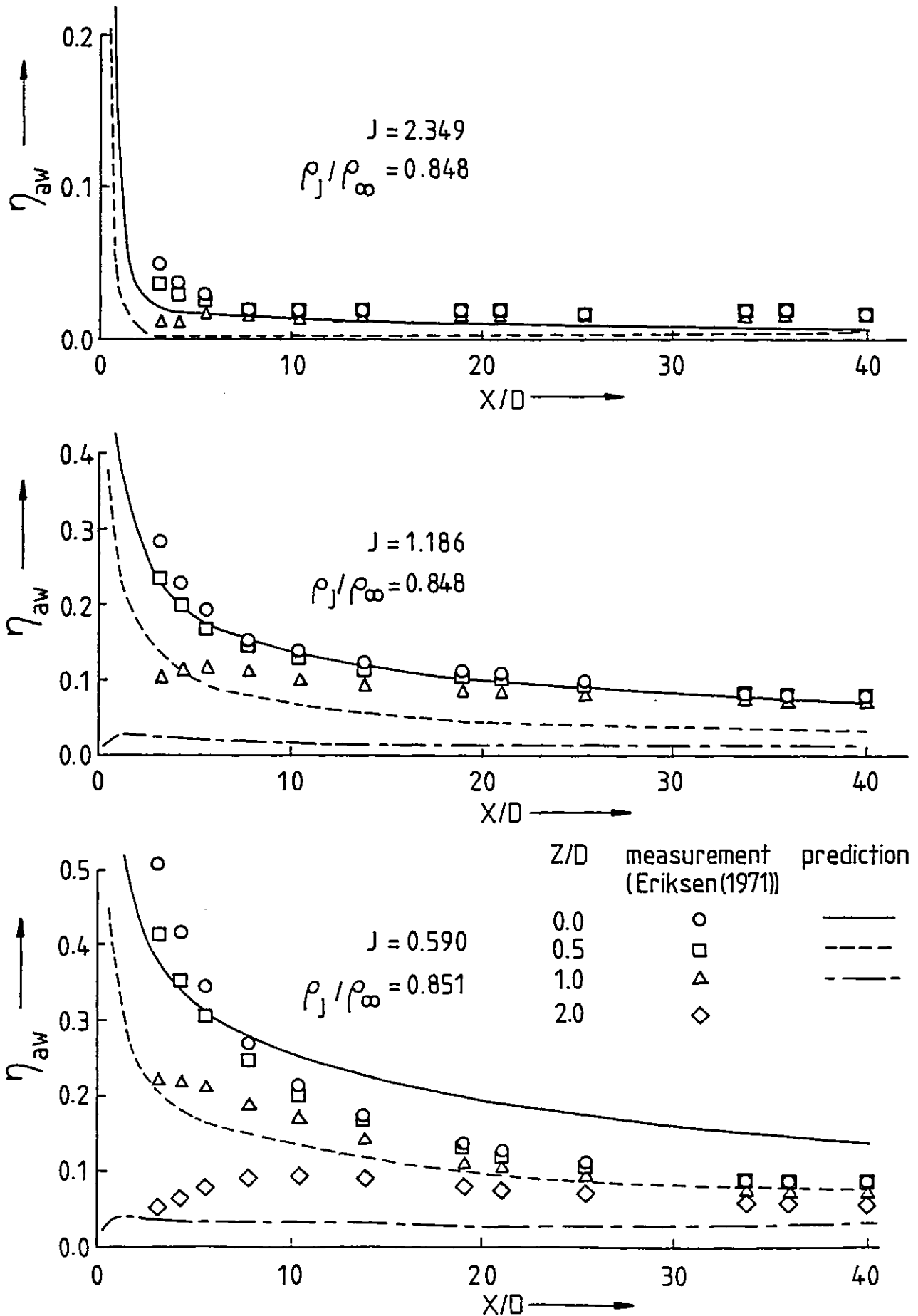


Fig. 6.12 Adiabatic wall effectiveness

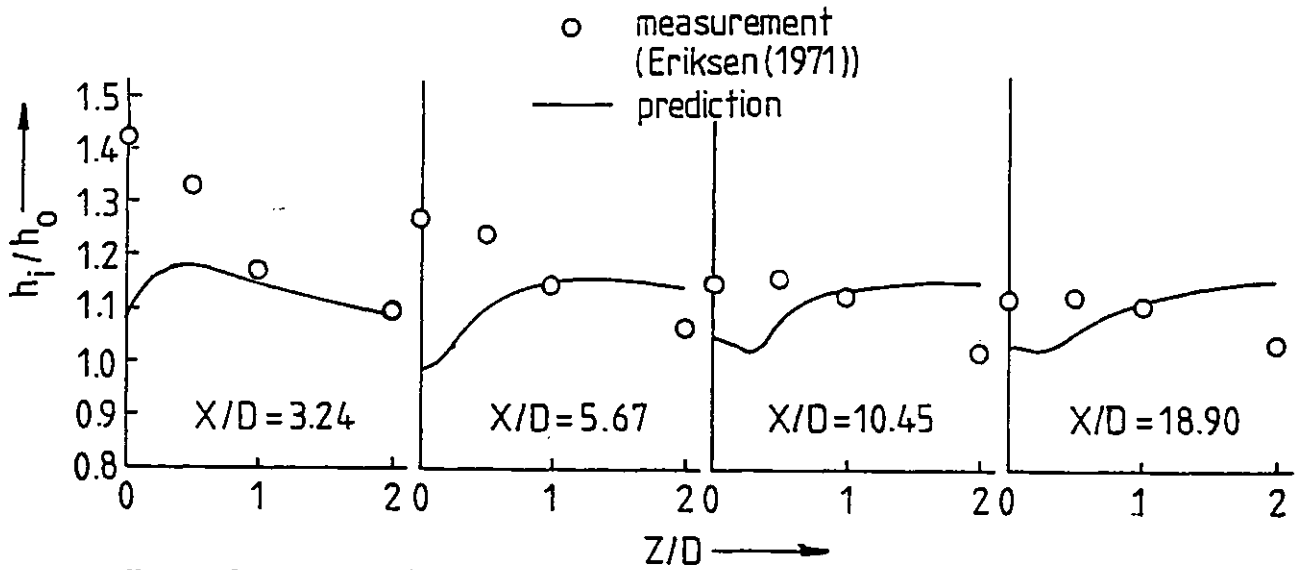


Fig. 6.13 Surface heat transfer coefficients, $J = 1.990$,

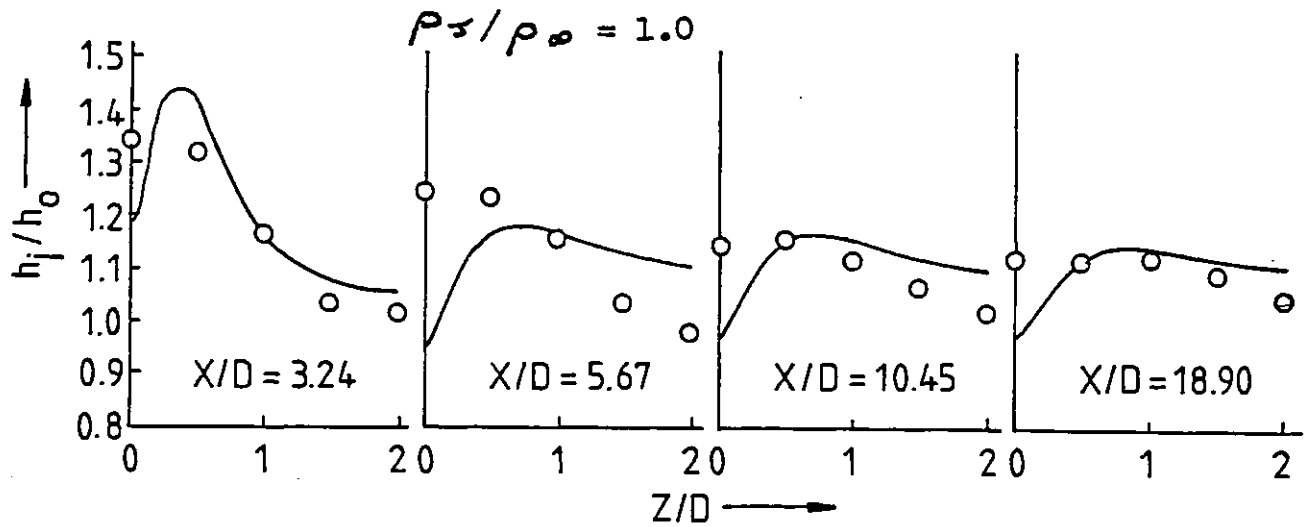


Fig. 6.14 Surface heat transfer coefficients, $J = 1.013$,

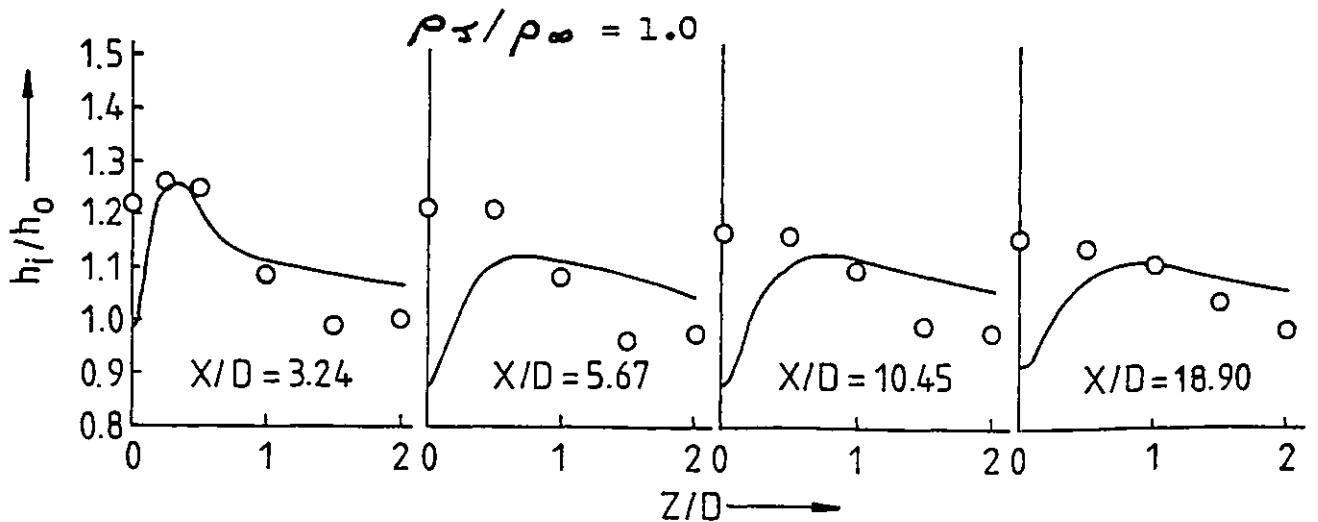


Fig 6.15 Surface heat transfer coefficients, $J = 0.544$,
 $\rho_s/\rho_\infty = 1.0$

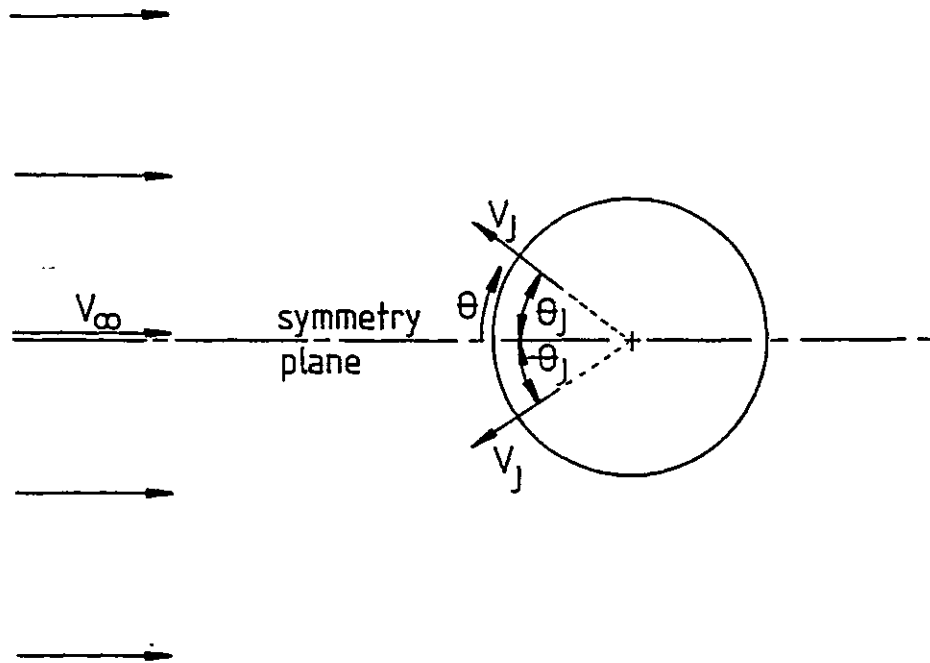


Fig. 7.1 Definition of plane of symmetry

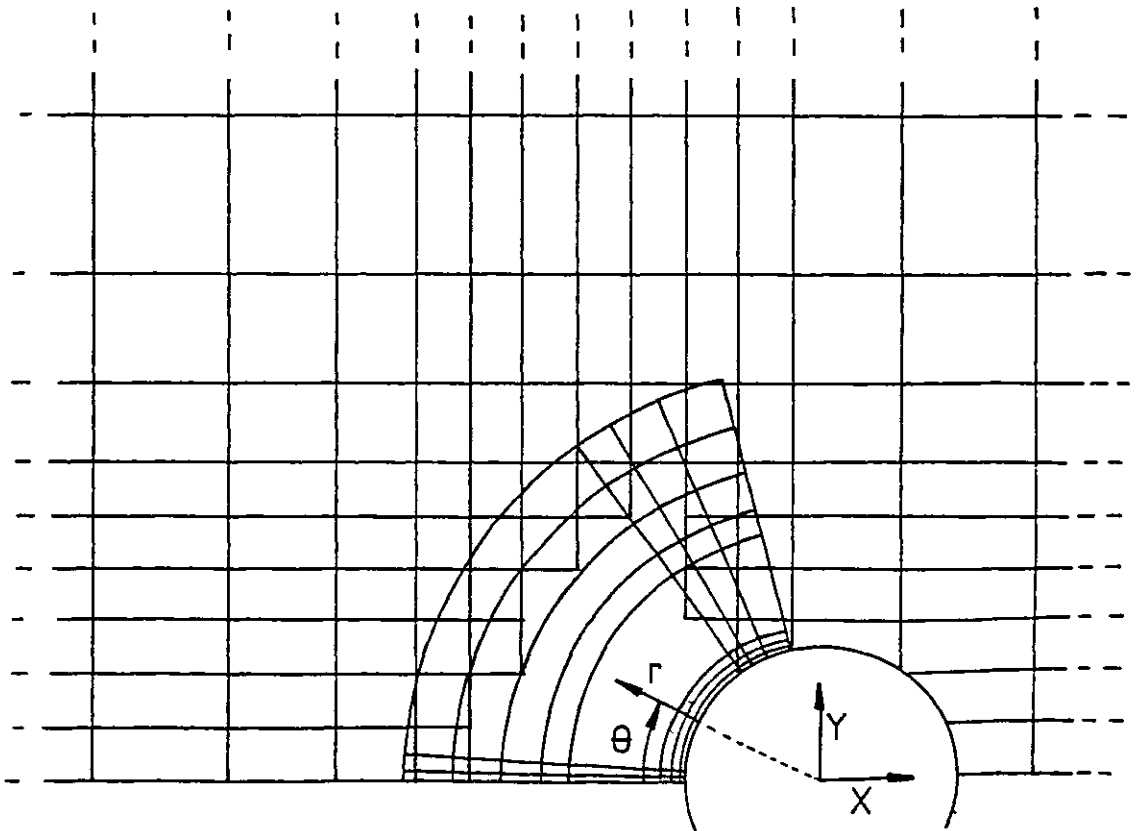


Fig. 7.2 Schematic diagram of embedded grid system

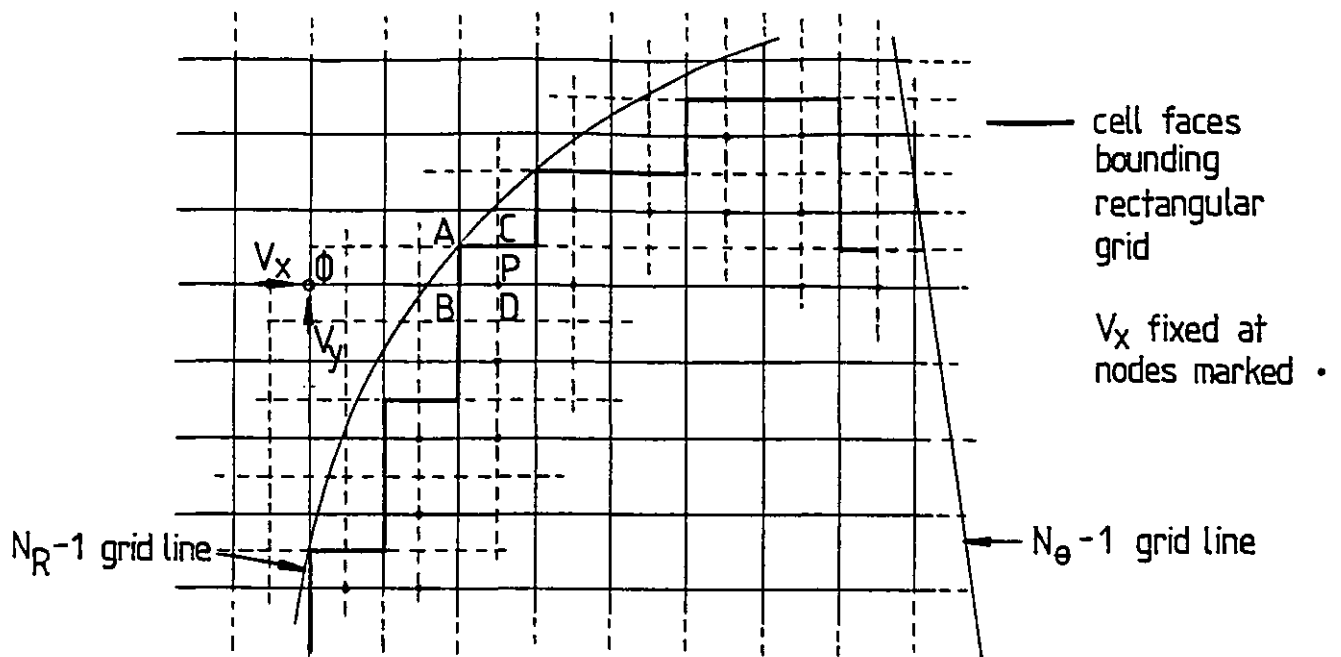


Fig. 7.3(a) Rectangular grid : Preliminary V_x - cell interface

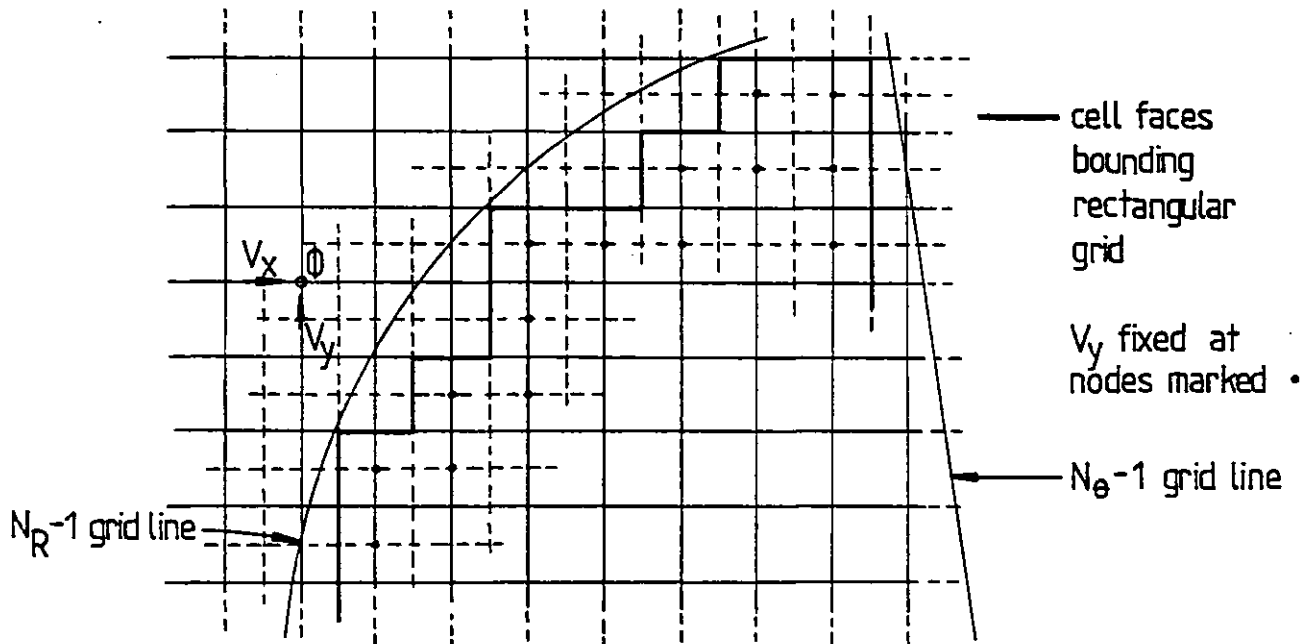


Fig. 7.3(b) Rectangular grid : Preliminary V_y - cell interface

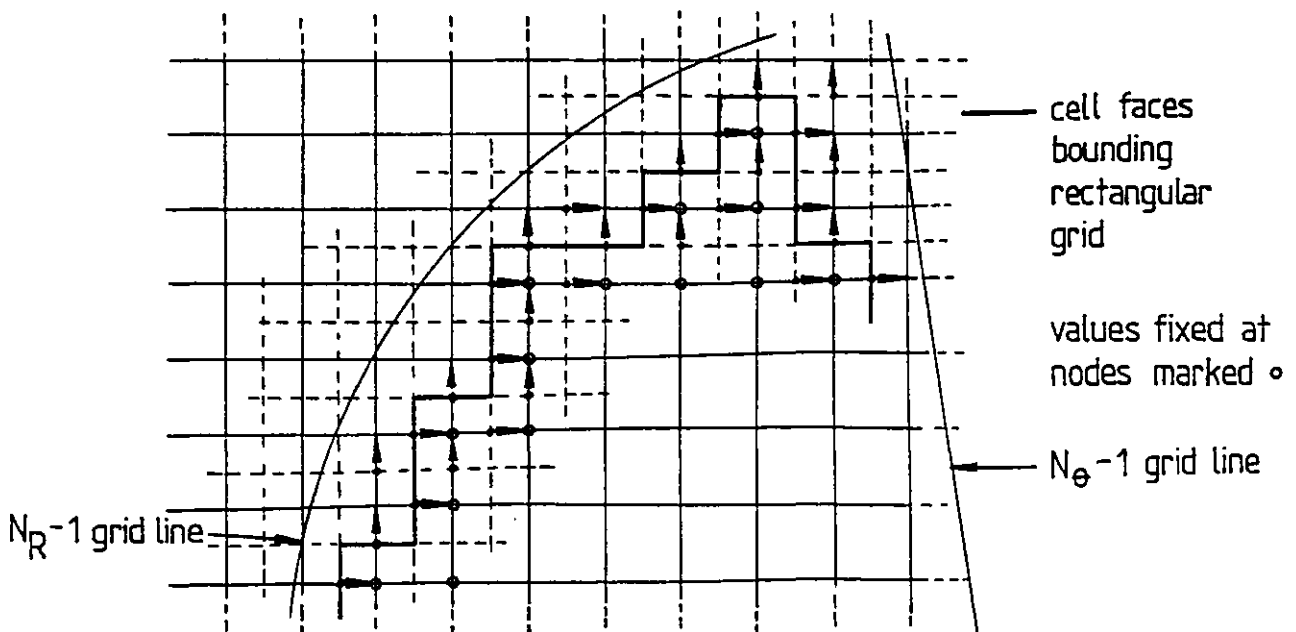


Fig. 7.3(c) Rectangular grid : V_x and scalar cell interface

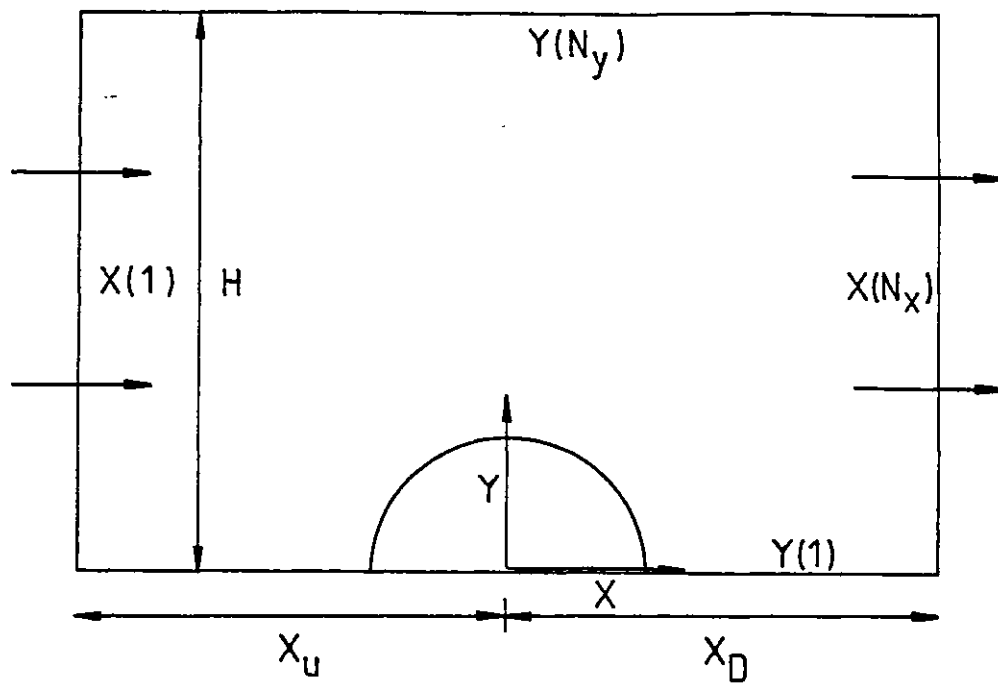


Fig. 7.4 Domain for rectangular grid laminar flow solutions

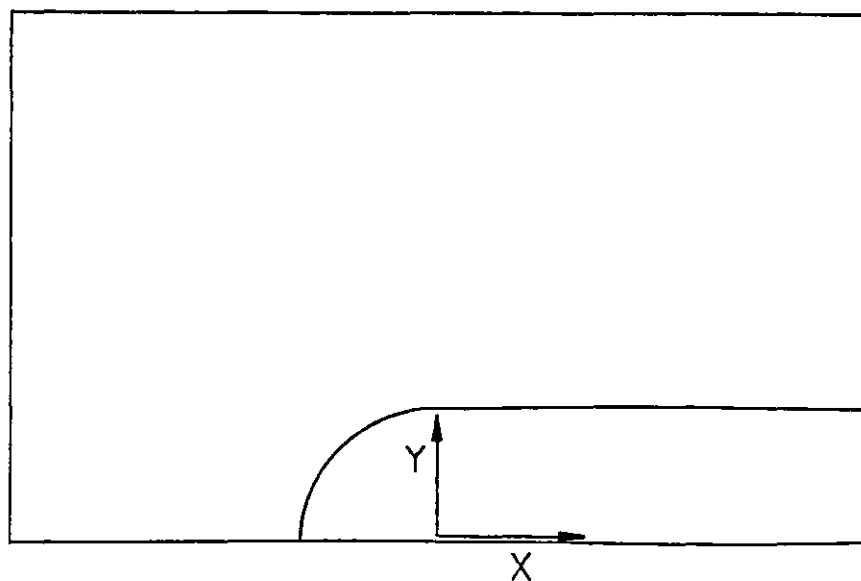


Fig. 7.5 Cylinder with afterbody

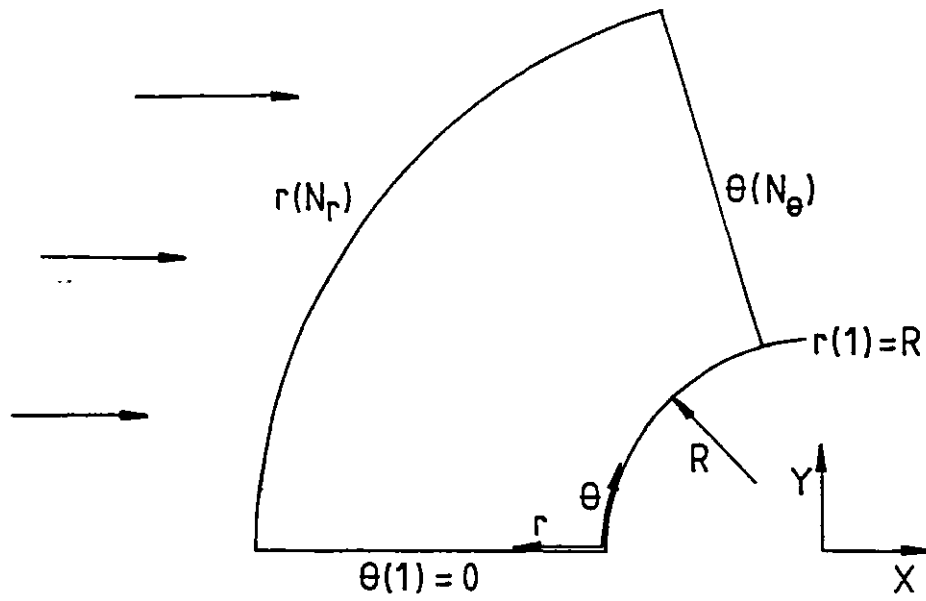


Fig. 7.6 Cylindrical grid solution domain in the $r-\theta$ plane

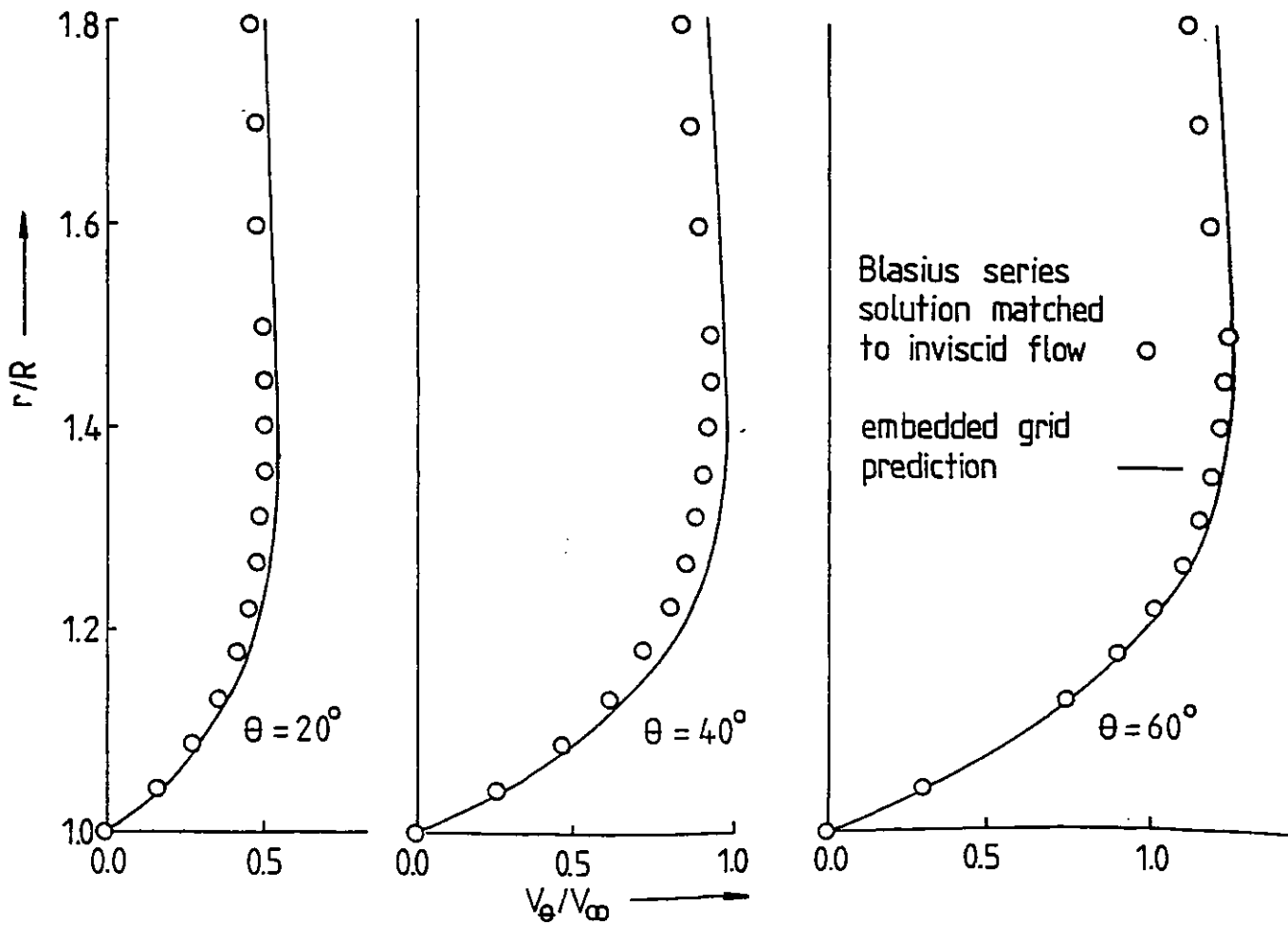


Fig. 7.7 V_θ - velocity profiles for $Re_D = 40$

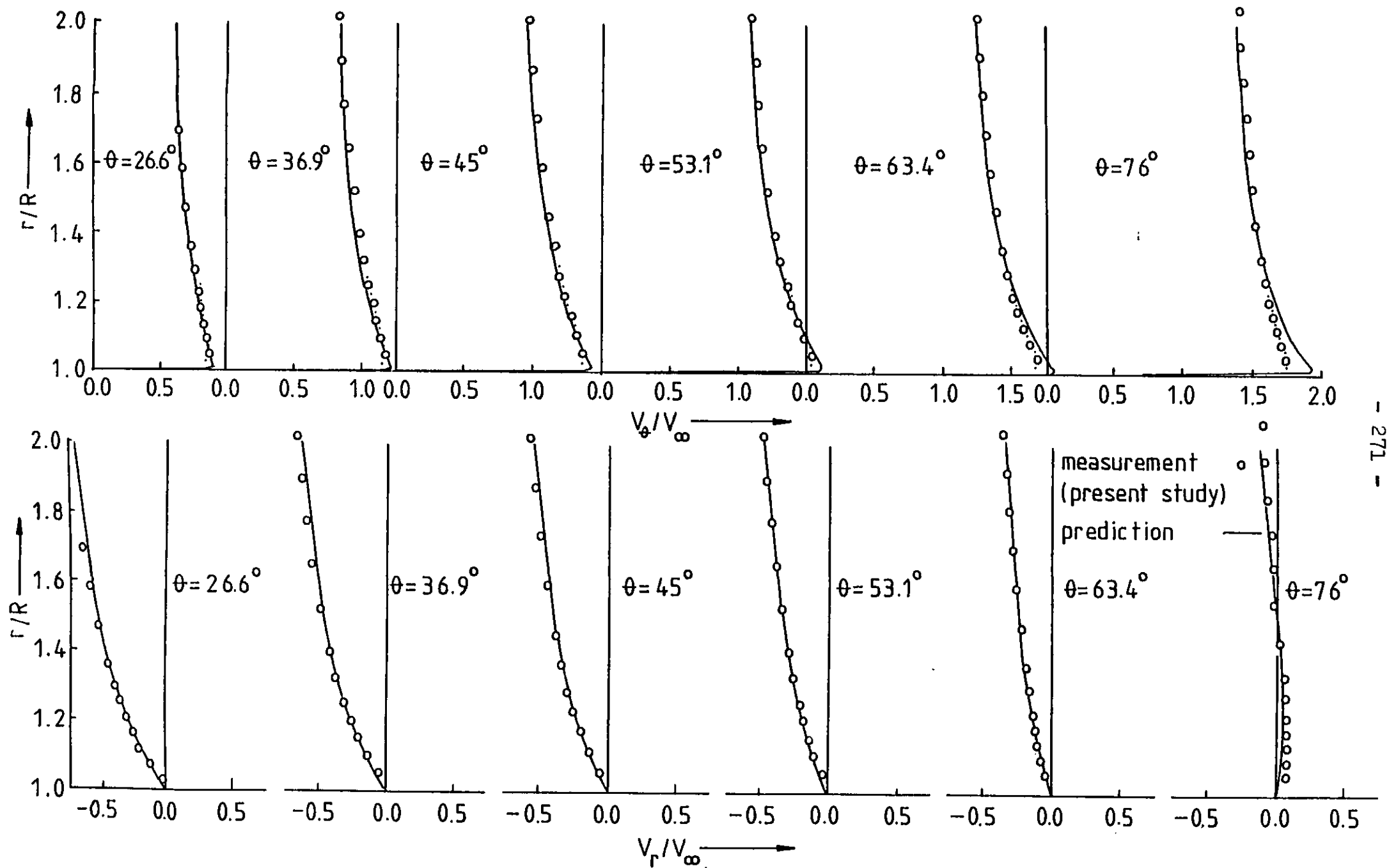


Fig. 7.8 Velocity profiles for cylinder without ejection

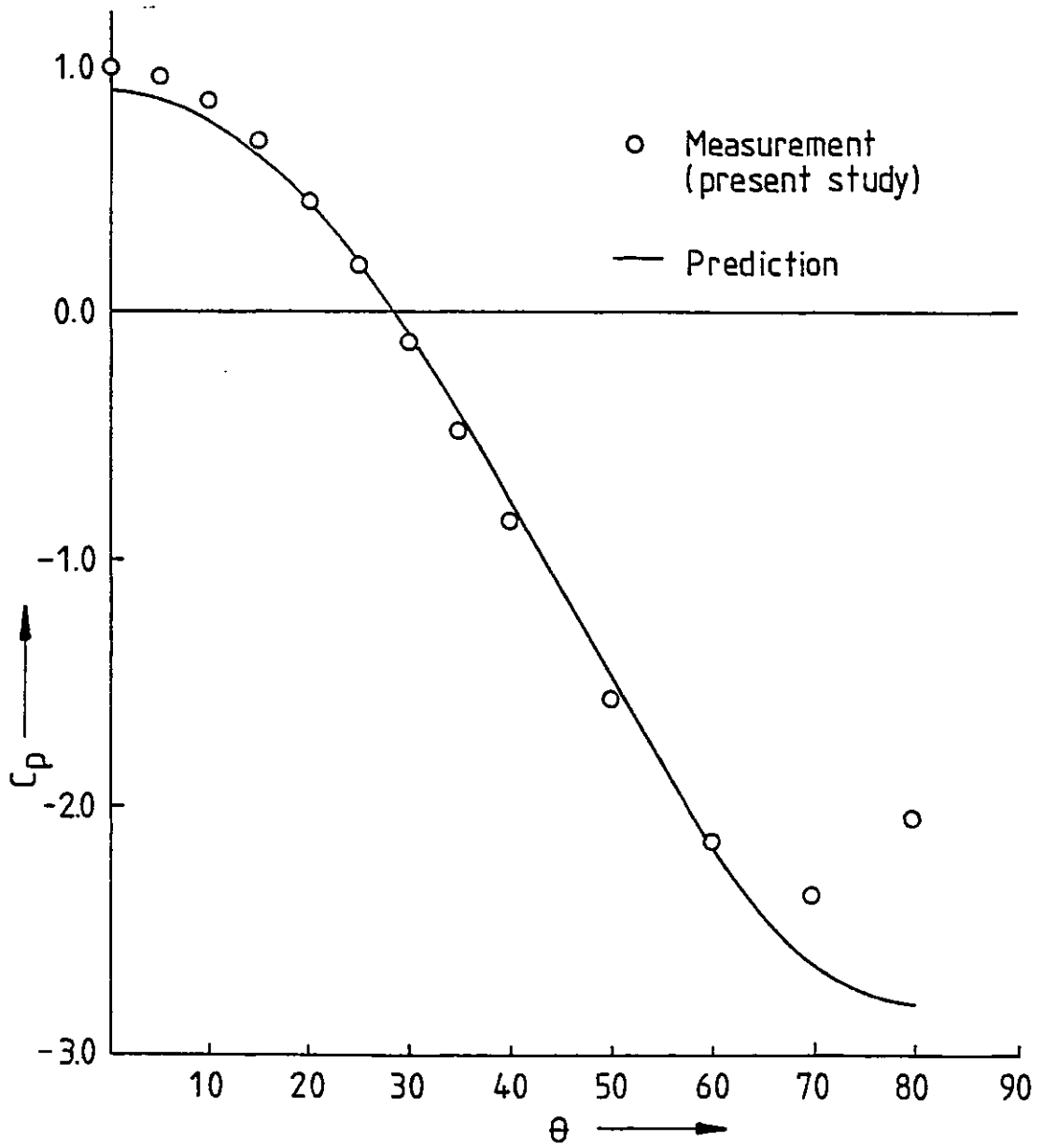


Fig. 7.9 Static pressure distribution on cylinder without ejection

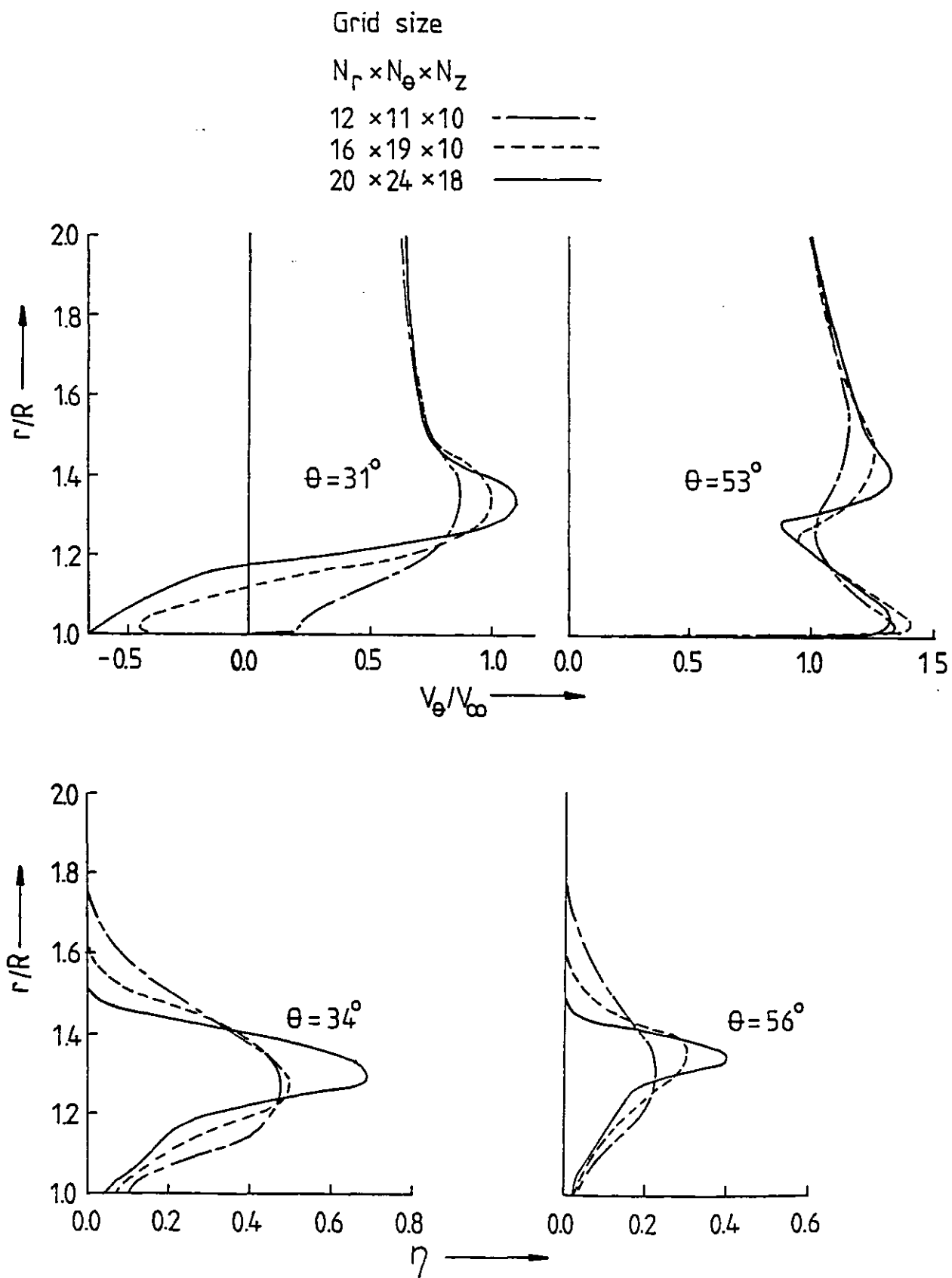


Fig. 7.11 Effect of grid refinement on the cylindrical grid solution, $\theta_\gamma = \pm 20^\circ$, $J = 2.0$, $Z/D = 0.0$

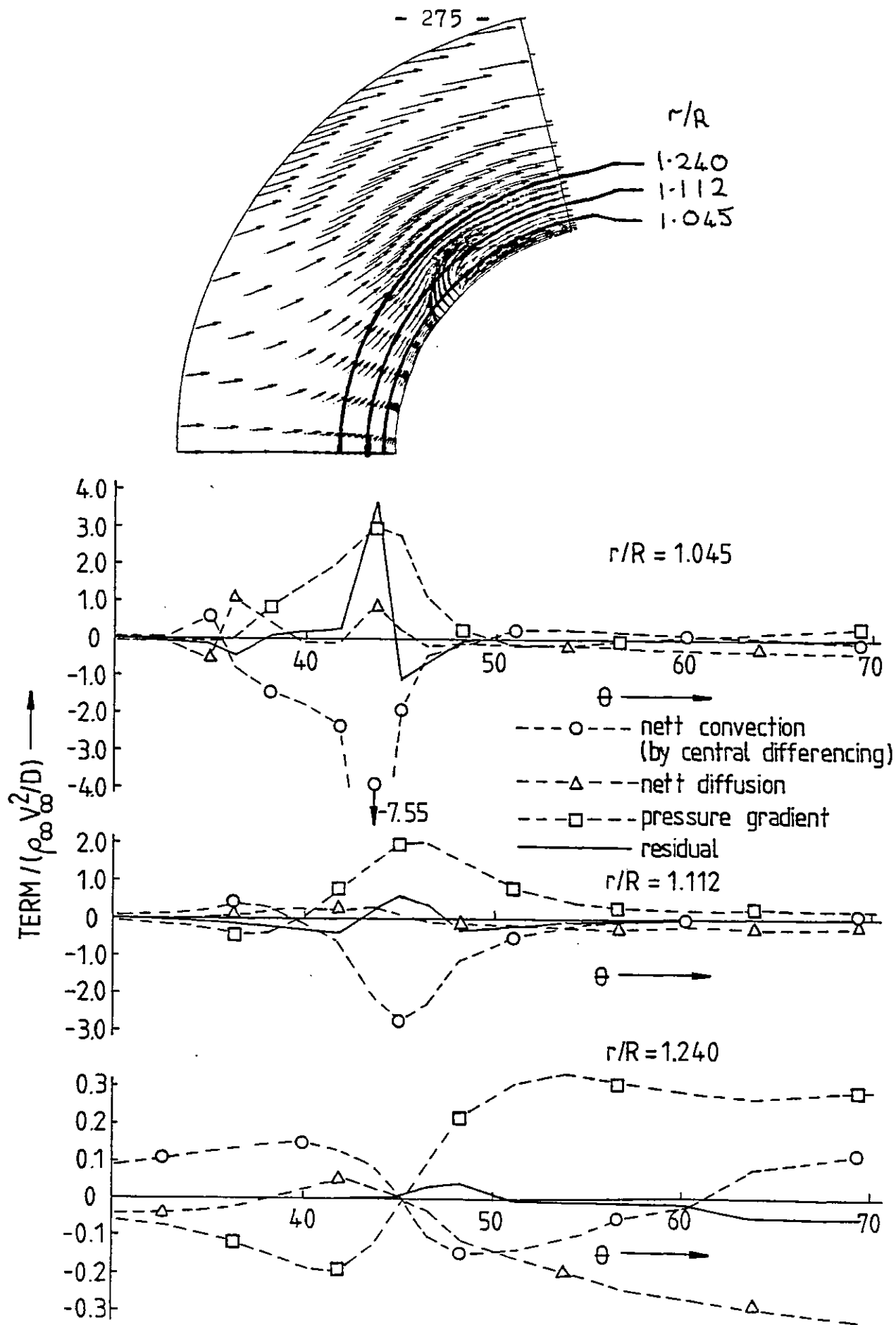


Fig. 7.12(a) Profiles of terms in the radial momentum equation,

$$\Theta_s = \pm 40^\circ, J = 1.81, z/D = 0.25$$

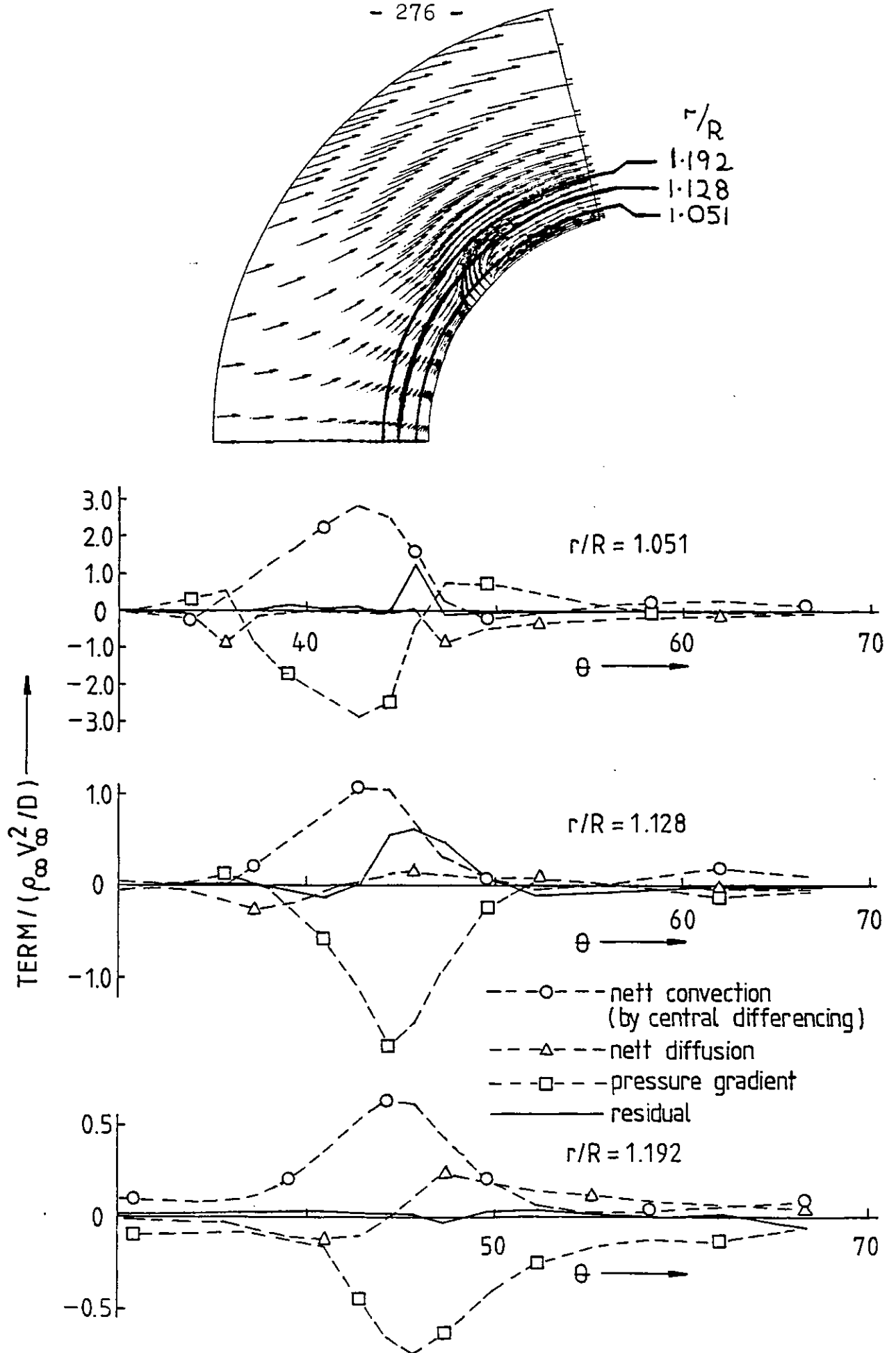


Fig. 7.12(b) Profiles of terms in the tangential momentum equation,
 $\theta_{\gamma} = \pm 40^{\circ}$, $J = 1.81$, $z/D = 0.25$

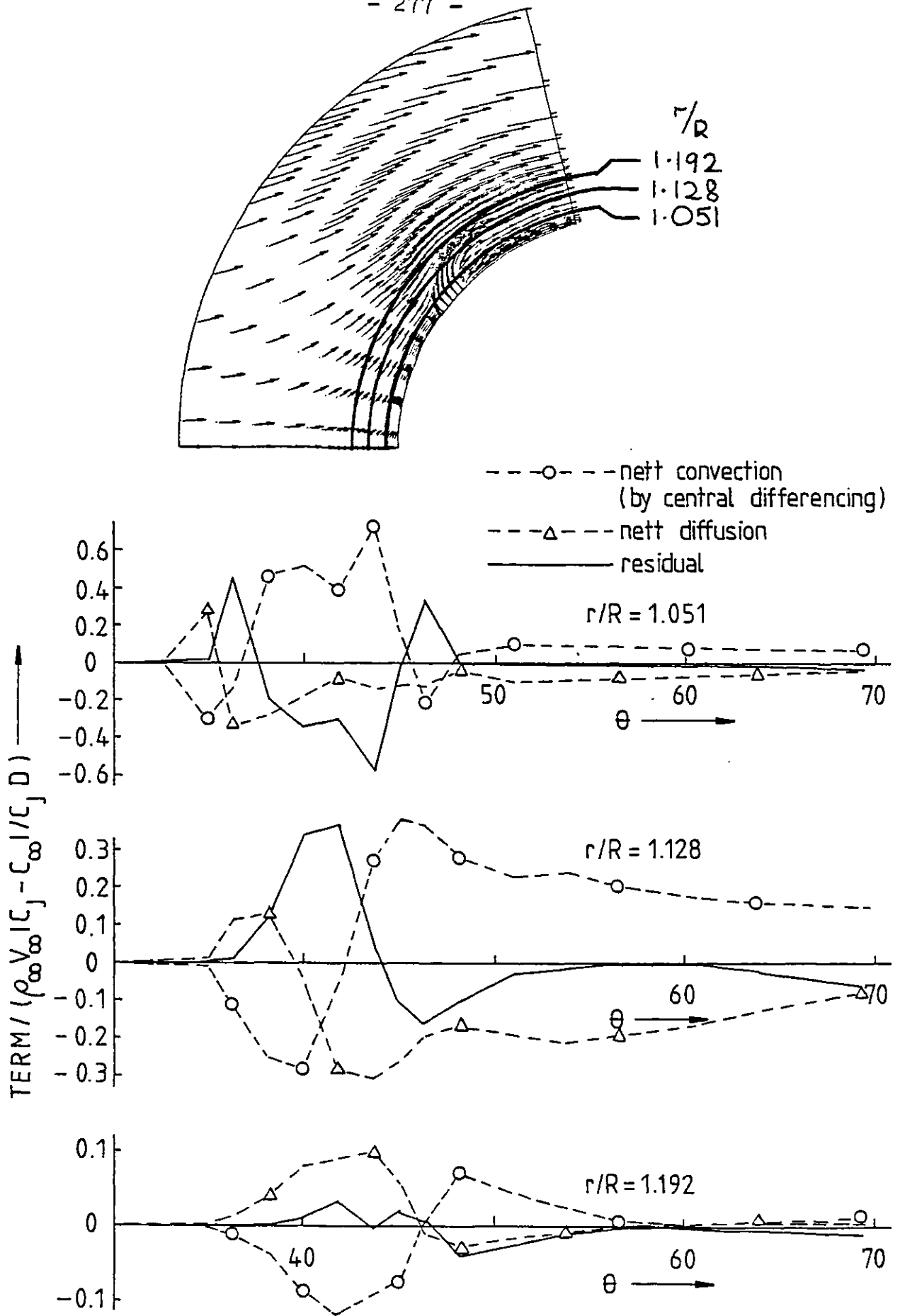


Fig. 7.12(c) Profiles of terms in the equation for the conserved scalar, C , $\theta_J = \pm 40^\circ$, $J = 1.81$, $z/D = 0.25$

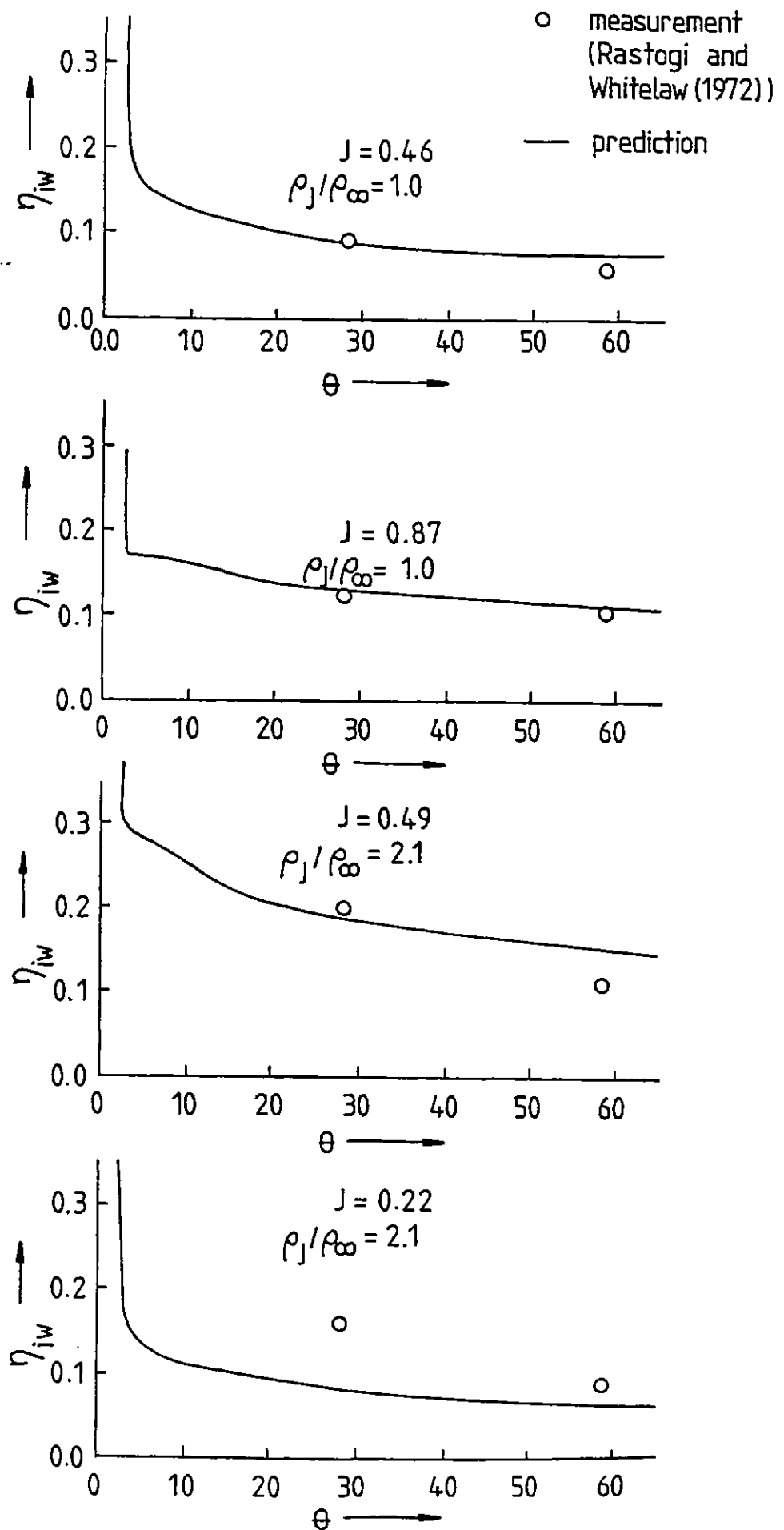
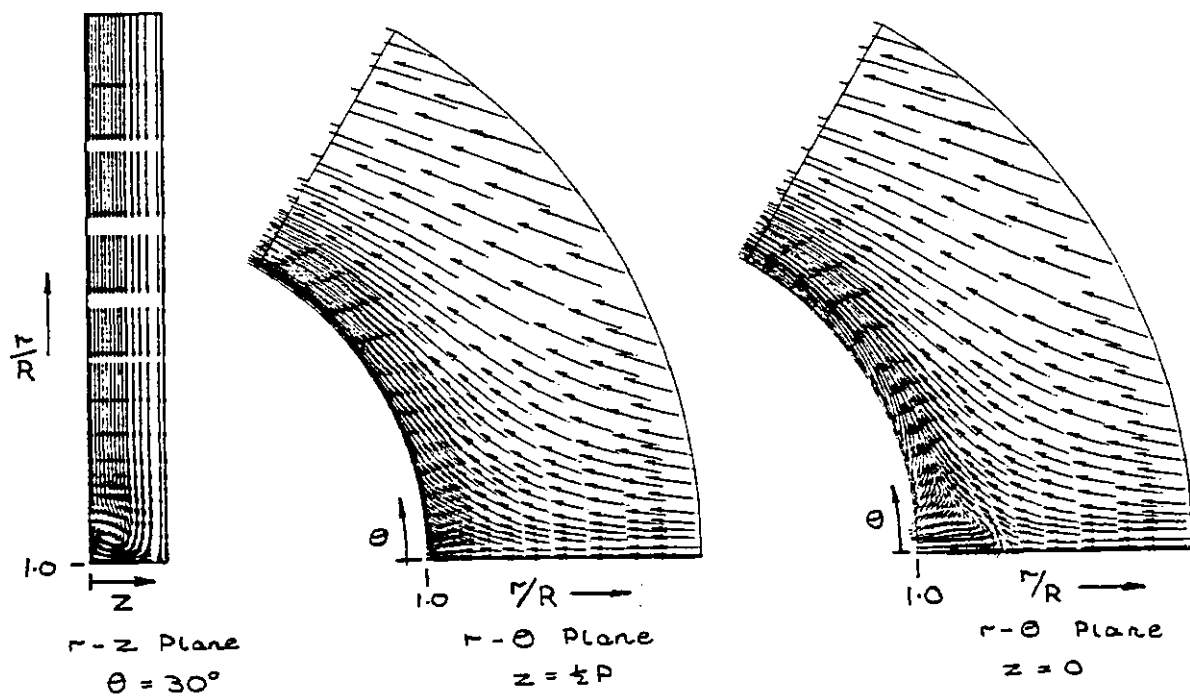
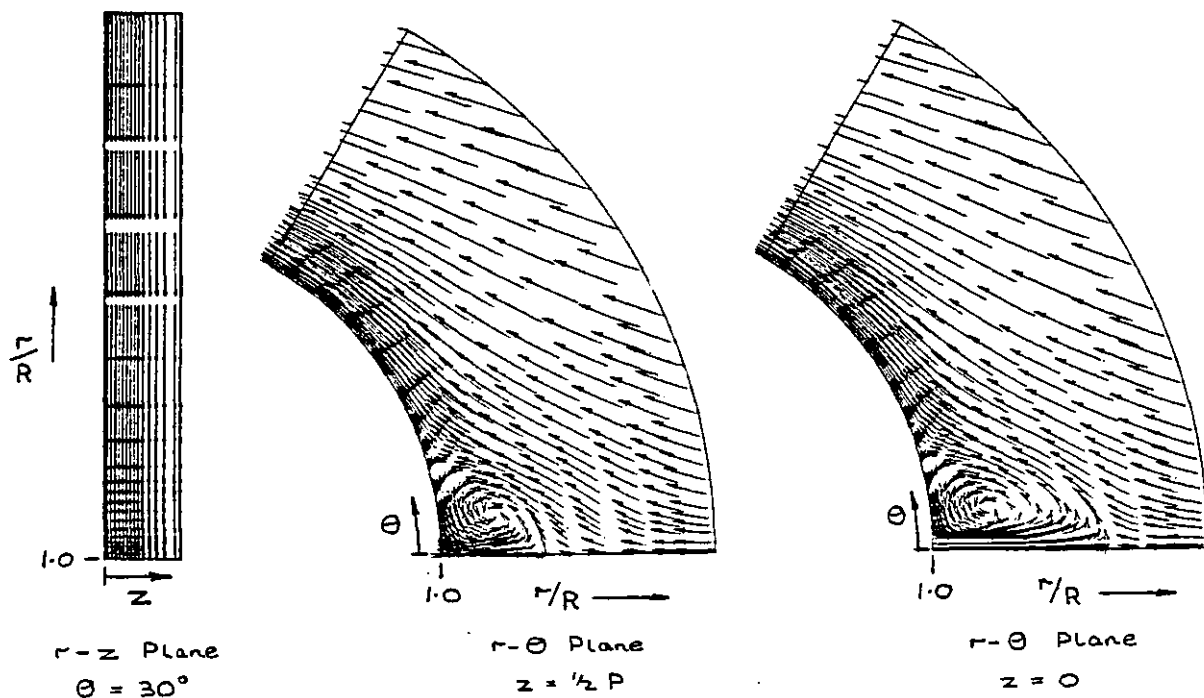


Fig. 7.13 Impervious wall effectiveness, $\Theta_r = 0^\circ$, $z/D = 0.0$



$$\rho_5 V_5^2 / \rho_\infty V_\infty^2 < 1.0$$



$$\rho_5 V_5^2 / \rho_\infty V_\infty^2 > 1.0$$

Fig. 7.14 Predicted flow regimes for $\Theta_5 = 0^\circ$

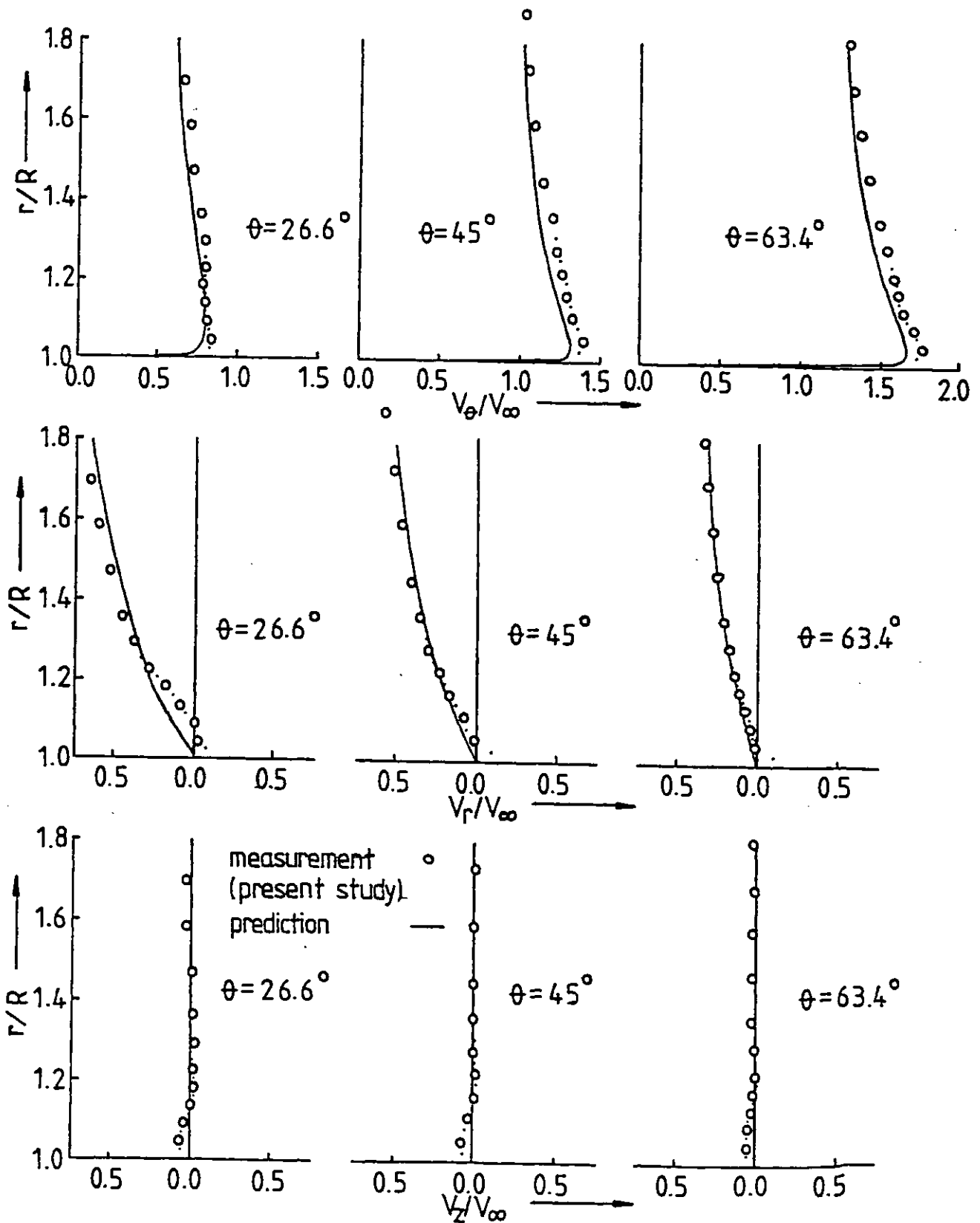


Fig. 7.15(a) Velocity profiles, $\theta_x = 0^\circ$, $J = 0.89$, $z/D = 0.0$

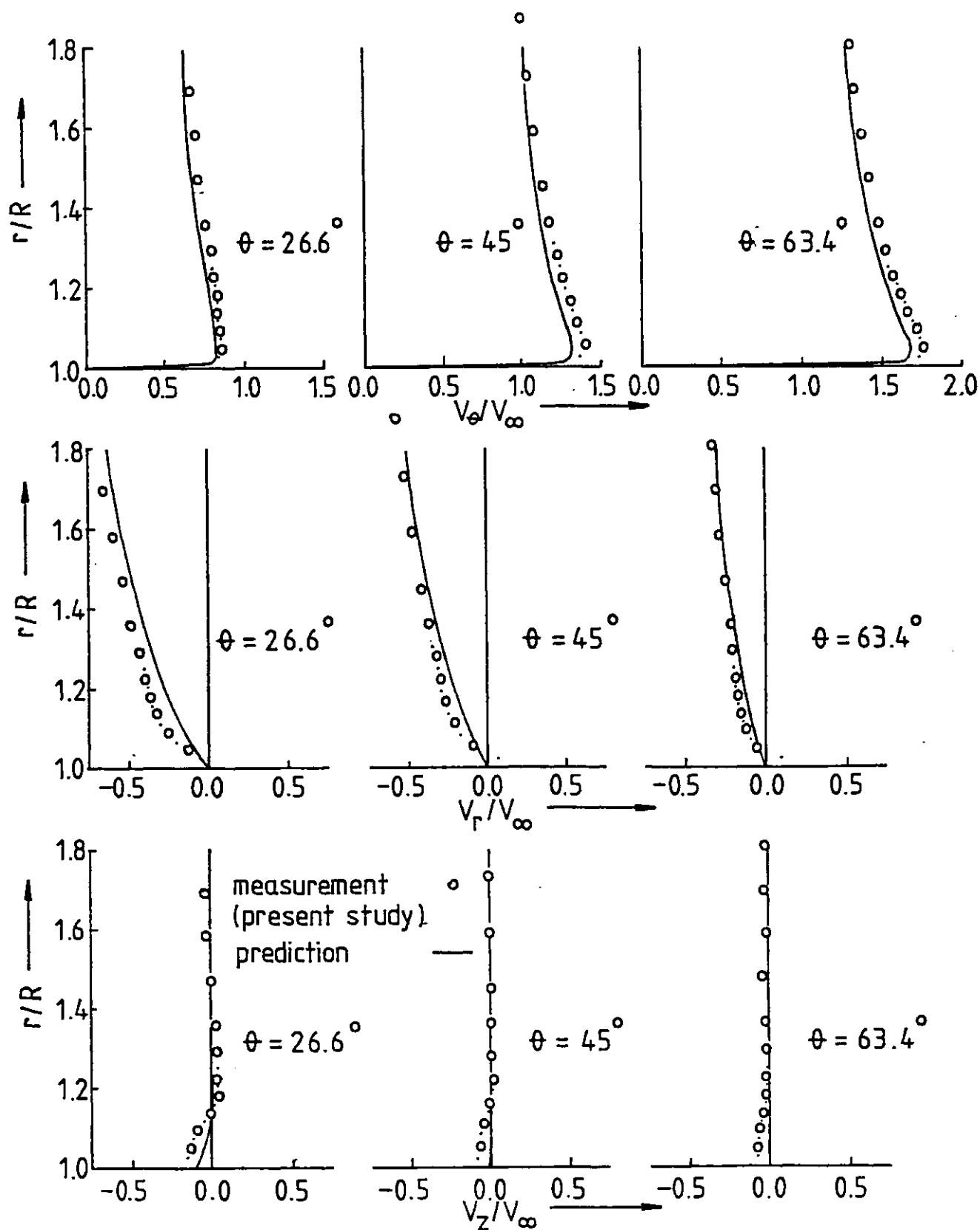


Fig. 7.15(b) Velocity profiles, $\Theta_r = 0^\circ$, $J = 0.89$, $z/D = 1.0$

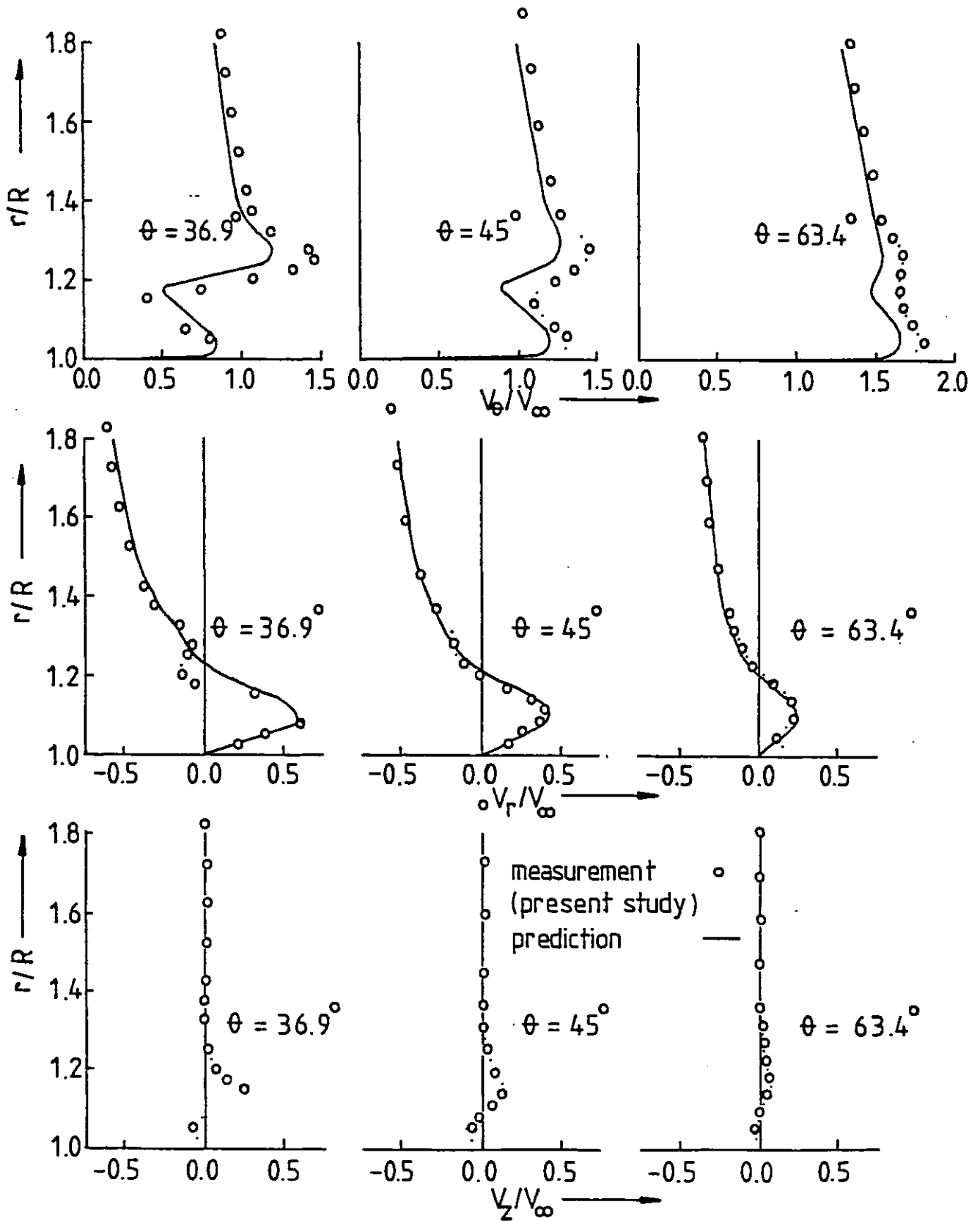


Fig.7.16(a) Velocity profiles, $\Theta_r = \pm 20^\circ$, $J = 1.98$, $Z/D = 0.0$

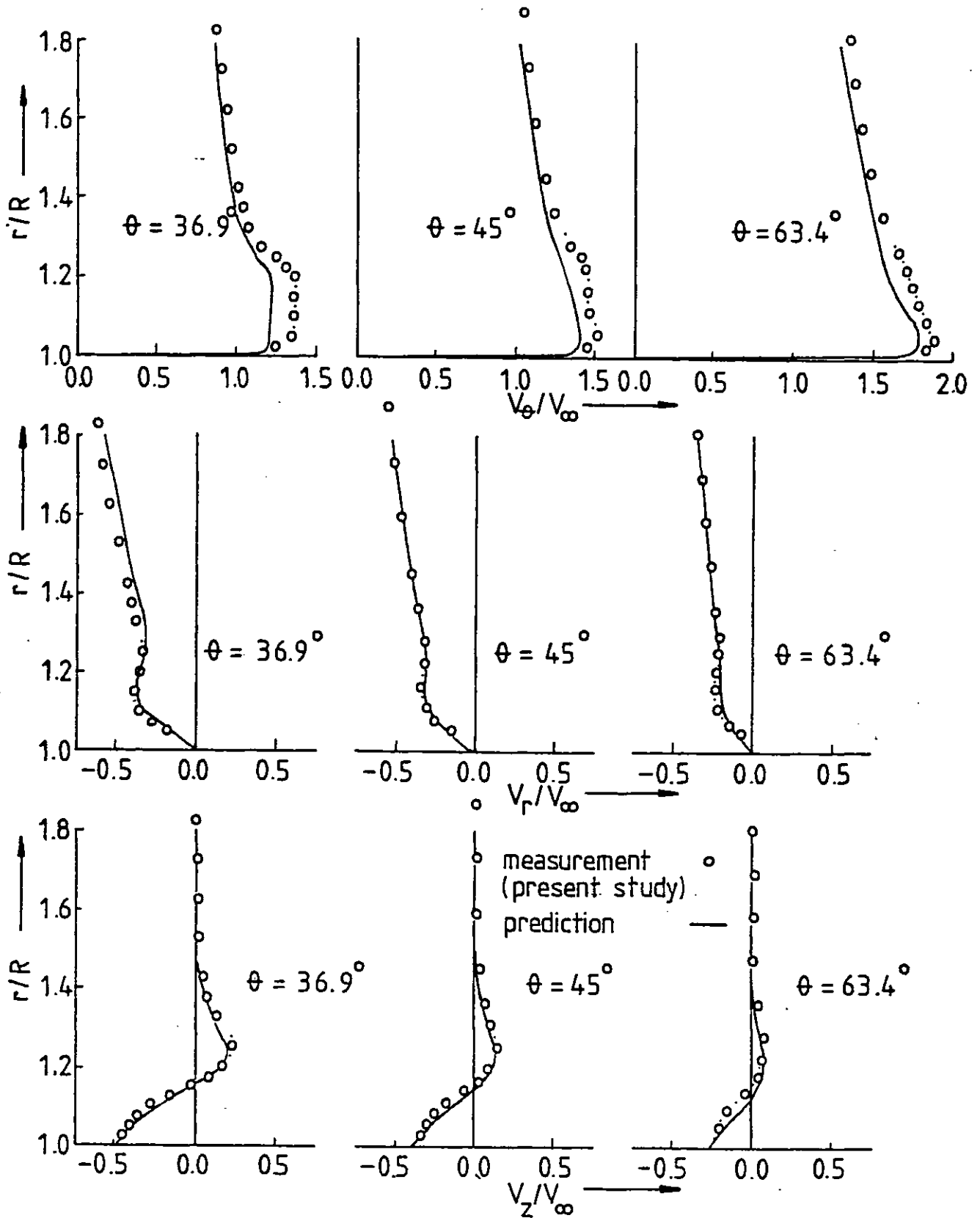


Fig. 7.16(b) Velocity profiles, $\Theta_\gamma = \pm 20^\circ$, $J = 1.98$, $Z/D = 1.0$

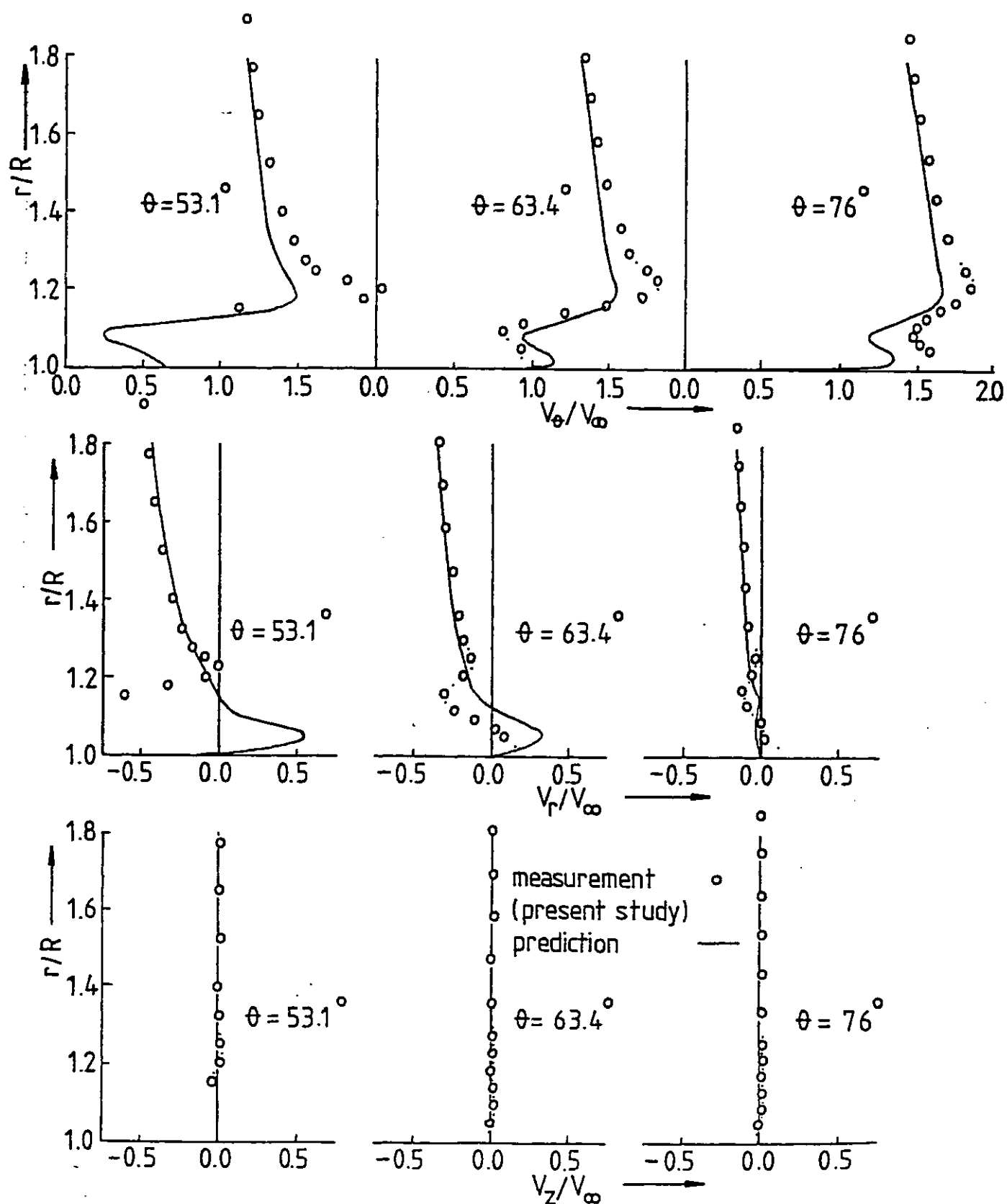


Fig. 7.17(a) Velocity profiles, $\Theta_\tau = \pm 40^\circ$, $J = 1.81$, $Z/D = 0.0$

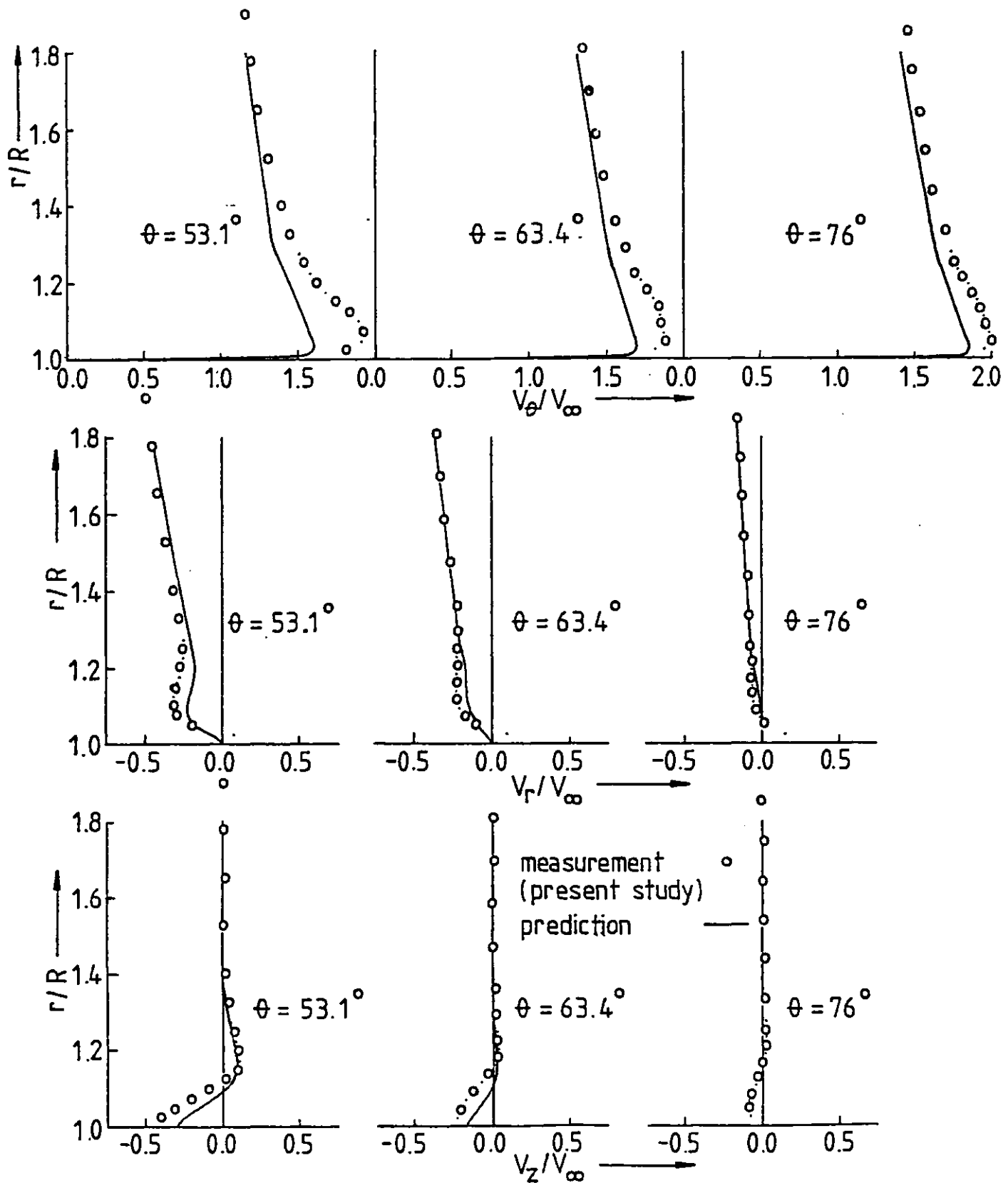


Fig. 7.17(b) Velocity profiles, $\Theta_T = \pm 40^\circ$, $J = 1.81$, $Z/D = 1.0$

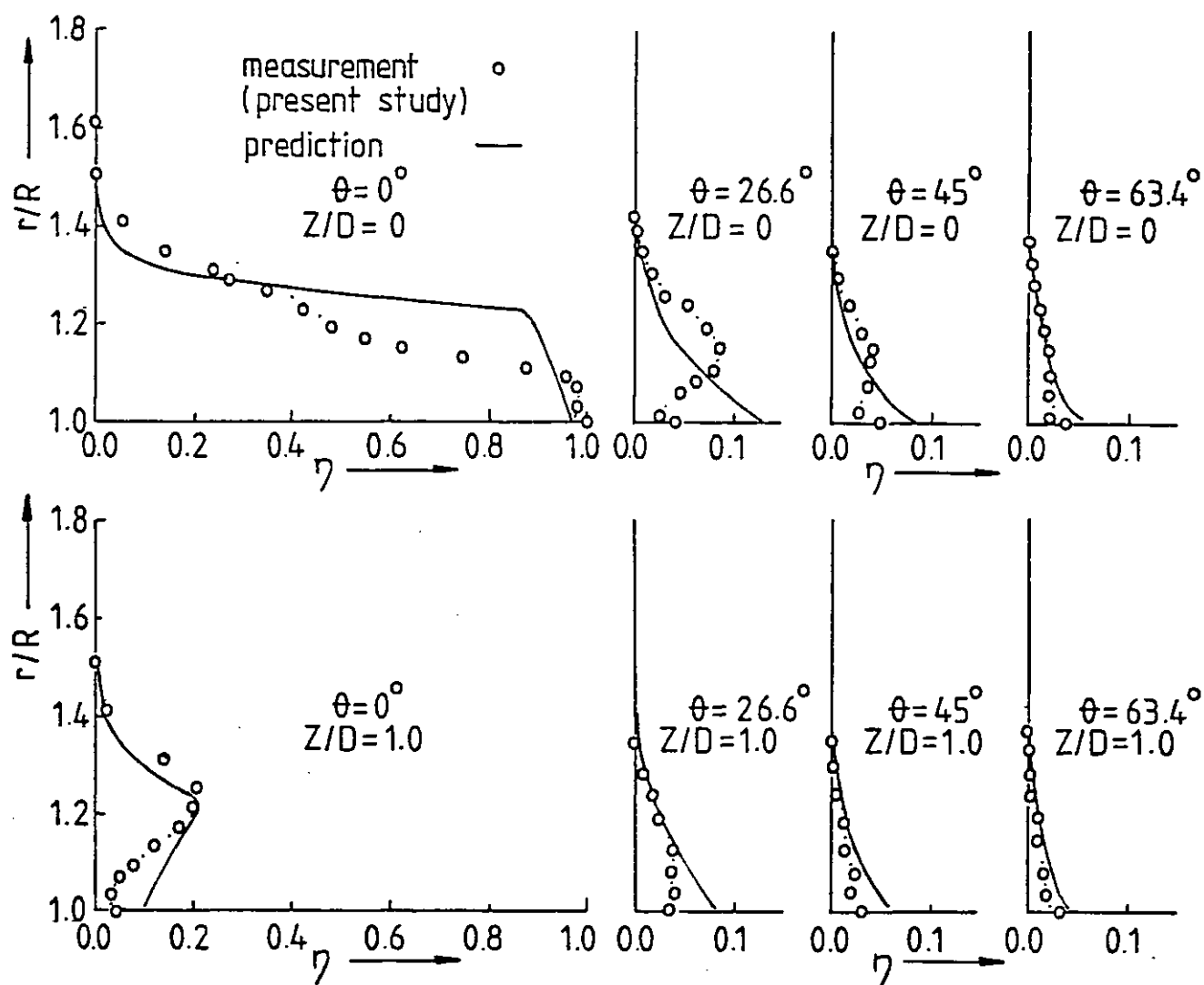


Fig. 7.18 Concentration profiles, $\Theta_J = 0^\circ$, $J = 0.89$

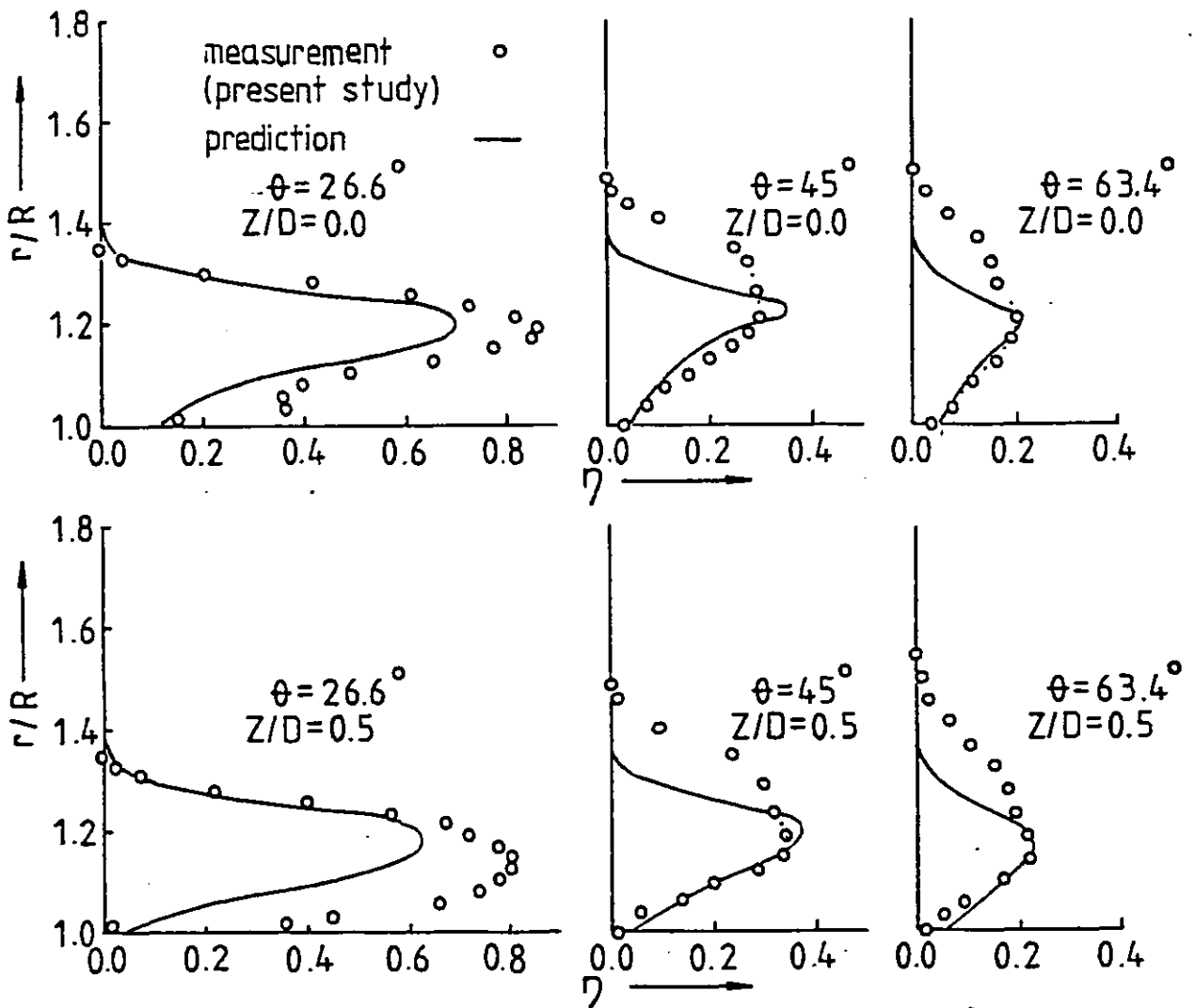


Fig. 7.19(a) Concentration profiles, $\theta_r = \pm 20^\circ$, $J = 1.98$

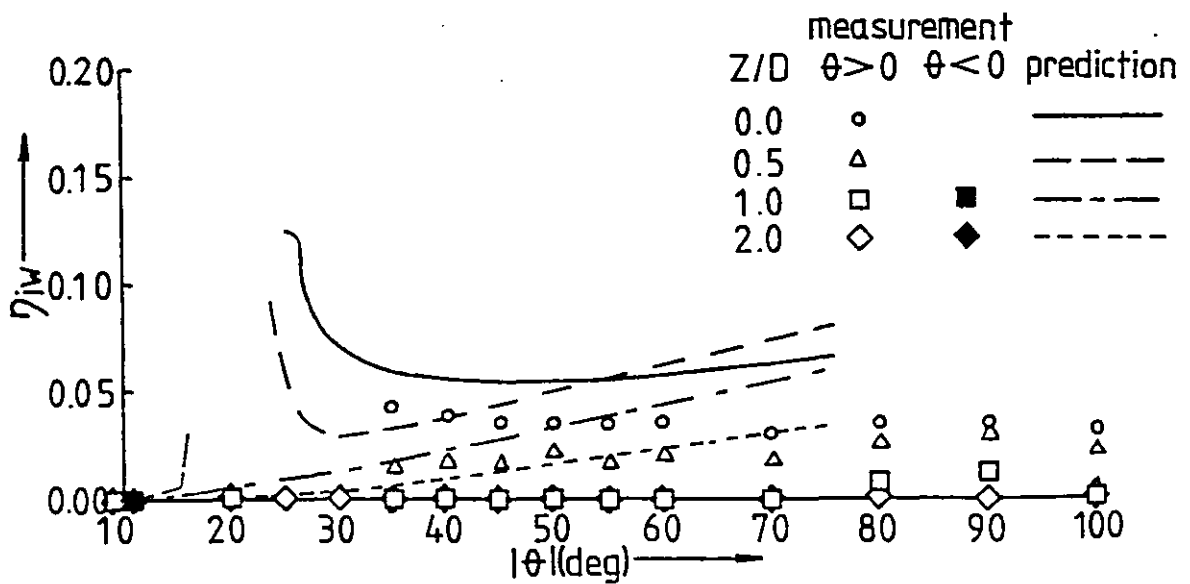


Fig. 7.19(b) Surface concentration, $\theta_r = \pm 20^\circ$, $J = 1.98$

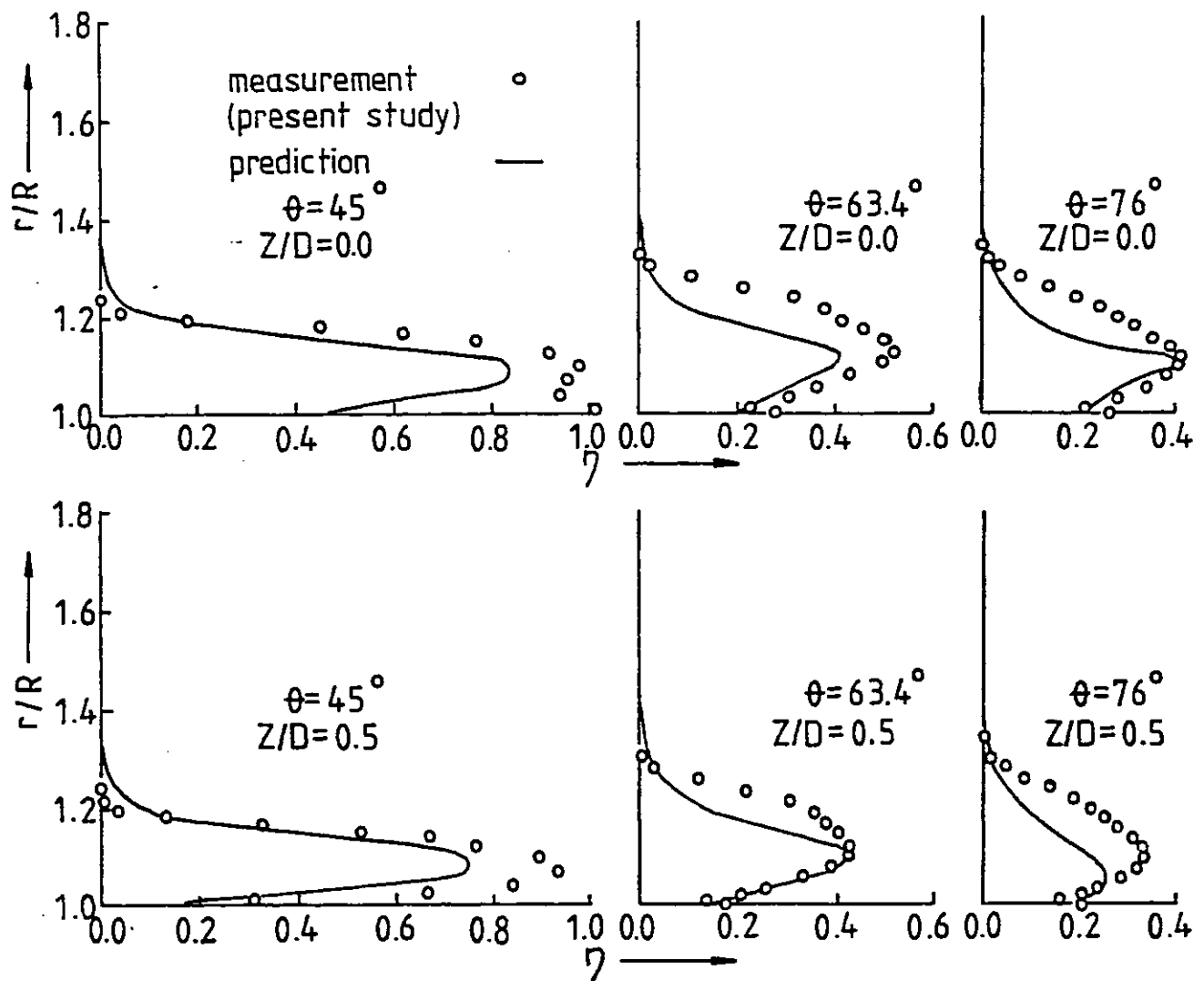


Fig. 7.20(a) Concentration profiles, $\theta_x = \pm 40^\circ$, $J = 1.81$

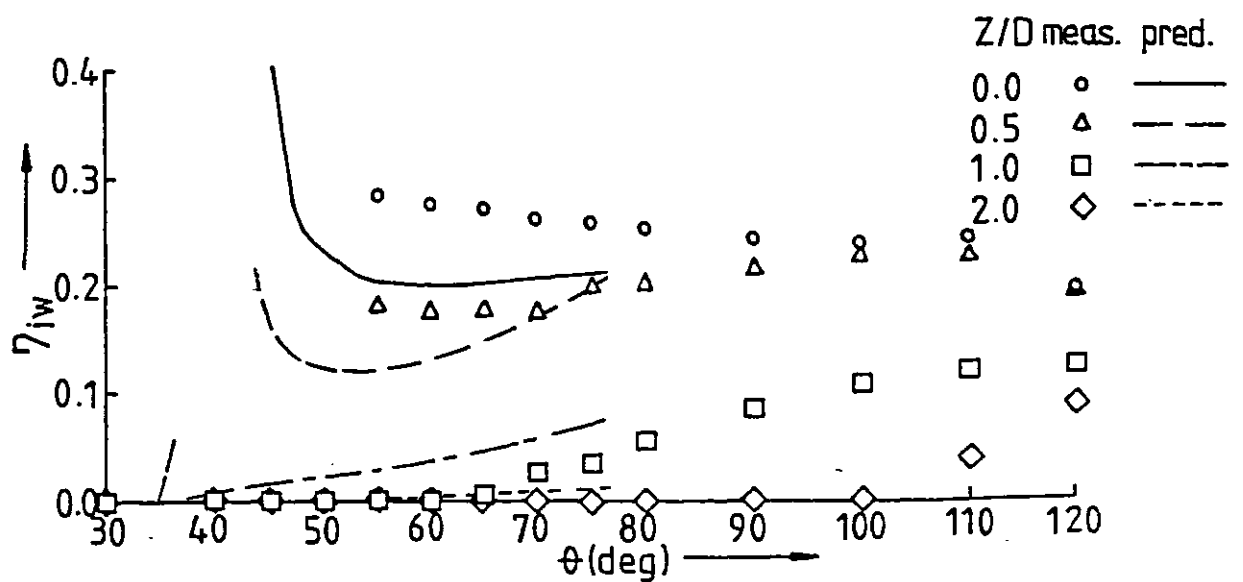


Fig. 7.20(b) Surface concentration, $\theta_x = \pm 40^\circ$, $J = 1.81$