

THREE-DIMENSIONAL DISCRETE-HOLE
COOLING PROCESSES.
AN EXPERIMENTAL AND THEORETICAL STUDY

by

GEORGIOS BERGELES

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

Mechanical Engineering Department

Imperial College

London SW7

July 1976

ABSTRACT

The thesis presents an experimental and theoretical investigation of 3-dimensional discrete hole cooling processes. The extensive experimental programme undertaken for various cooling geometries (single hole, single row and double row) was aimed at obtaining details of the flow and mixing in the vicinity of the injection holes, including the upstream region. These regions have been omitted, or received only slight attention in earlier studies of discrete-hole cooling processes. The experimental study sheds light on the comparative efficiency of the various cooling geometries and the effect of the parameters, boundary layer thickness, blowing rate and angle of injection on film cooling effectiveness. The study shows the existence of a blowing ratio (M_{opt}) which gives maximum cooling coverage, and is a function of the boundary-layer thickness and cooling geometry. For blowing ratios less than M_{opt} the injected jet remains attached to the surface, provoking considerable pressure and velocity disturbances and giving rise to increasing effectiveness as M is increased. At values of M greater than M_{opt} , the jet lifts off the surface, allowing penetration of the mainstream fluid beneath it. As a consequence the effectiveness now diminishes as M is augmented, as are the surface pressure disturbances and a complex structure is produced. A further conclusion is that a double row of holes geometry performs better than a single row one with the same amount of coolant injected into the boundary layer.

The theoretical part of the thesis describes the application of a 3-dimensional finite-difference procedure to the problem of

predicting the flow and thermal fields arising in discrete-hole cooling configurations. The developed prediction procedure is validated against experimental data obtained in the present experimental study and by other workers. The procedure predicts very satisfactorily the flow field in discrete-hole cooling geometries.

The numerical scheme employed in the solution of the 3-dimensional parabolic equations is of the semi-elliptic type: this entails that, while the pressure field is held in 3-dimensional arrays, the velocity field is stored only 2-dimensionally, the velocities of the whole flow field being obtained by marching in the streamwise direction.

The present work proposed and employed a non-isotropic ($k - \epsilon$) turbulence model in which the diffusion coefficients in the two directions on the cross-stream planes are proportional to the corresponding normal stresses.

Finally a specific treatment has been developed for incorporating reverse flow regions within the framework of semi-elliptic type of flows and an improvement of this treatment is suggested for solving flow fields predominantly parabolic which contain small regions of strong reverse flow.

Acknowledgments

I would like to express my sincere thanks to those persons who contributed to the completion of the present work.

First and foremost I am indebted to my Supervisors, Drs. B. E. Launder and A. D. Gosman for their sound guidance during my research work and their constructive criticism in the writing of the thesis. Our regular Monday meetings in reappraising the week's progress are highly appreciated for the scientific scrutiny to which the work was subjected. I regard it a privilege to have worked with them so closely.

In the initial stages of the work discussions with Dr. V. S. Pratap on the semi-elliptic procedure and various comments from Prof. D. B. Spalding on the essence of the semi-elliptic approach are acknowledged with thanks.

At that time, the numerical explorations were based on a version of the commercial 3-dimensional parabolic program STABL (suitably modified to include the semi-elliptic feature) kindly loaned to me by CHAM Ltd. Later, I developed a Departmental program with equivalent capabilities starting from the fully-elliptic TEACH-3E code developed under Dr. Gosman's supervision by Mr. J. A. C. Humphrey and Mr. N. Vlachos.

The help offered by Dr. M. M. Ribeiro in using his complex hot-wire anemometry set-up is appreciated.

Mr. J. Pilmer responsible for the laboratory, Mr. A. Robinson for keeping in good condition the various instruments and Mr. L. Varty for ordering the materials needed have greatly assisted with their prompt cooperation in the successful completion of the experimental programme.

IV

Thanks are due also to the staff of the Mechanical Engineering Department workshop for constructing the various test plates.

I express my gratitude to Dr.A. Morris from the NGTE and Mr. B. Barry from Rolls-Royce (1971) Ltd. for the simulating research meetings we held and the interest they took in the project.

Thanks are also due to Miss Denise Basing for typing most of the thesis.

Numerical computations were performed with the help of computers at the Imperial College Computer Center, the University of London Computer Center and the SRC's Rutherford Computer Laboratory. My thanks are due to the staff at all these Institutions for their helpfulness.

The financial support for the research has been generously provided by the Ministry of Defence, Procurement Executive (NGTE) and Rolls Royce (1971) Ltd.

	Page	
ABSTRACT	I	
ACKNOWLEDGMENTS	III	
CONTENTS	V	
LIST OF TABLES AND FIGURES	XI	
CHAPTER 1	INTRODUCTION	1
1.1	Background	1
1.2	A review of previous studies	6
1.2.1	Interaction of a jet with a cross-flow	7
1.2.2	Correlation formulae for the effectiveness of discrete hole cooling systems	10
1.2.3	Experimental studies of discrete hole cooling phenomena	13
1.3	Outline of the present study	33
1.3.1	Objectives of the present research programme	33
1.3.2	Layout of the thesis	
CHAPTER 2	APPARATUS AND MEASUREMENT TECHNIQUES	36
2.1	Introductory remarks	36
2.2	Apparatus	36
2.2.1	The wind tunnel	36
2.2.2	The secondary air supply system	38
2.2.3	The test plates	38
2.3	Measurement techniques	39
2.3.1	Flow visualisation techniques	39
2.3.2	Quantitative measurements	40

VI

		<u>Page</u>
2.4	Accuracy of the measurements	44
2.5	Reproducibility of the measurements	45
2.6	Operating procedures and experimental conditions	46
2.6.1	Operating procedures	46
2.6.2	Experimental conditions	
CHAPTER 3	PRESENTATION AND DISCUSSION OF THE MEASUREMENTS	47
3.1	Injection through a single hole	47
3.1.1	Injection through a single hole normal to the mainstream flow	47
3.1.2	Injection through a single hole at 30°	54
3.2	Injection through a single row of holes at 30°	61
3.3.	Injection through a double row of holes at 30°	68
3.4	Conclusions of the present experimental study	74
3.4.1	Hydrodynamic field	74
3.4.2	Concentration field	75
CHAPTER 4	THEORETICAL STUDY	77
4.1	The mathematical foundation of the problem	77
4.1.1	Introduction	77
4.1.2	The equations	77
4.1.3	The equations for a quasi-orthogonal co-ordinate system	78

VII

	<u>Page</u>
4.1.4 Some simplifications for the discrete- hole cooling (DHC) geometry	80
4.1.5 The simplified form of the equations on the (ξ, η, ζ) co-ordinate system	83
4.1.6 Initial and boundary conditions	85
4.2 The mathematical modelling of turbulence	87
4.2.1 Introductory remarks	87
4.2.2 The steady state momentum, energy and continuity equations for turbulent flows	89
4.2.3 The closure problem	89
4.2.4 The modelling of the Reynolds stresses	90
4.2.5 The modelling of the turbulent heat fluxes	91
4.2.6 Assessment of the turbulence model proposed for the present study	92
4.2.7 The inlet and boundary conditions for K, ϵ	94
4.2.8 Summary	95
4.3 The numerical solution of the equations	96
4.3.1 Introduction	96
4.3.2 The procedure for discretizing the differential equations	98
4.3.3 The "SIMPLE" algorithm	105
4.3.4 The treatment of the pressure field	107
4.3.5 The pressure correction equation for parabolic and semi-elliptic flows	108

VIII

		<u>Page</u>
4.3.6	Summary of the "SIMPLE" algorithm	110
4.3.7	The wall numerical boundary conditions	111
4.3.8	The relative merits of the parabolic and semi-elliptic procedure	112
4.3.9	Some other aspects of the numerical procedure	113
CHAPTER 5	VALIDATION OF THE PREDICTION PROCEDURE	
5.1	Introductory remarks	115
5.2	Preliminary calculations	118
5.2.1	Grid-dependence tests	118
5.2.2	Effect of the velocity boundary condition over the hole	118
5.2.3	Effect of the non-isotropic viscosity model	119
5.3	Some computational characteristics of the semi-elliptic procedure	120
5.3.1	The rate of convergence	120
5.4	Presentation of the computations: single hole	121
5.4.1	Injection through a single hole 90° to the mainstream flow	121
5.4.2	Injection through a single hole 30°	128
5.5	Injection through a single row of holes	135
5.5.1	Measurements by Eriksen, $\alpha=35^{\circ}$, M=0.2	135
5.5.2	Measurements by Eriksen, $\alpha=35^{\circ}$, M=0.5	136

	<u>Page</u>
5.5.3 Measurements by Pedersen, $\alpha=35^\circ$, M=0.5	136
5.5.4 IC measurements, $\alpha=30^\circ$, M=0.25	137
5.5.5 IC measurements, $\alpha=30^\circ$, M=0.50	138
5.5.6 Some further characteristics of the flow field	139
5.6 Injection through a double row of holes	142
5.6.1 Introductory remarks	142
5.6.2 Measurements by Jabbari, $\alpha=35^\circ$, M=0.334	143
5.6.3 Measurements by Jabbari, $\alpha=35^\circ$, M=0.47	144
5.6.4 IC measurements, $\alpha=30^\circ$, M=0.25	145
5.6.5 Some further characteristics of the field	147
5.7 Concluding remarks	150
CHAPTER 6 CONCLUDING REMARKS	154
6.1 Introduction	154
6.2 A summary and assessment of the present investigation	154
6.2.1 Experimental investigation	154
6.2.2 Theoretical investigation	155
6.3 Recommendations for future work	158
6.3.1 Improvements to the computational procedure	158
6.3.2 Improvements to the turbulence model	162
6.3.3 Future applications of the comput- ational procedure	163

	<u>Page</u>
6.3.4 Extension of the experimental study	164
REFERENCES	166
NOMENCLATURE	171
APPENDIX A Error analysis of the velocity measurements downstream of the in- jection hole using total and static pressure probes	173
APPENDIX B Some detailed features of the computational procedure	178
APPENDIX C The prediction of discrete-hole cooling with the parabolic comput- ational procedure	182
APPENDIX D The prediction of three-dimensional discrete-hole cooling processes : I-Laminar flow	188
FIGURES	198

LIST OF TABLES AND FIGURES

	<u>Page</u>
1. Table 1 : Summary of experimental studies	15
2. Table 2 : Summary of the experimental conditions of the present study	37
3. Table 3 : Summary of initial and boundary conditions	85
4. Table 4 : S_{Φ} and Γ_{Φ} for each dependent variable Φ	99
5. Table A1: Estimation of the experimental errors	176
6. Chapter 1 : Figs. 1 - 12	198 - 204
7. Chapter 2 : Figs. 13 - 19	204 - 210
8. Chapter 3 : Figs. 20 - 56	210 - 248
9. Chapter 4 : Figs. 57 (a - g)	249 - 251
10. Chapter 5 : Figs. 58 - 89	252 - 298

CHAPTER 1

INTRODUCTION

1.1 Background

In recent years there has been growing interest in the behaviour of jets of fluid discharged obliquely into a mainstream flow. The basic flow configuration is to be found in such practical situations as the dispersion of effluent from chimney stacks, the discharge of hot water from power stations into lakes or rivers, the injection of secondary air into combustion chambers and the cooling of leading stages of gas turbine rotors and stators. In the first two examples, the desired outcome is the dispersion of the jet fluid as rapidly and completely as possible into the mainstream. In the last example however, - that of turbine blade cooling - the opposite outcome is sought, the main requirement being that the jets should not penetrate into the mainstream but stay as close as possible to the blade surface for thermal protection.

The demand for cooling the turbine blades arises from continuous increase of the turbine entry temperature in order to achieve higher specific thrust (engine thrust per unit of mass air flow) in the civil and military aero gas turbine engines; however the material of the blade cannot withstand such high temperatures and has to be cooled in order to achieve the required life under the operational centrifugal and thermal stress levels. Figure 1, taken from Fullagar's study (1), shows the turbine entry temperature for a series of Rolls-Royce engines. It is seen that over a period of 25 years the level of turbine entry temperature has almost doubled and that for temperatures above about 1000 K the blades require cooling. The doubling of the entry temperature over this period was accomplished by employing more

complicated and more efficient blade cooling methods in conjunction with improvements in the high-temperature properties of the materials of the blade.

Fullagar (1) gives the following indicative design picture for the appropriate cooling system versus turbine entry temperature:

Up to 1200 K	No cooling is necessary
Between 1200 K and 1450 K	Conventional internal convective systems will generally suffice
Between 1450 K and 1600 K	Conventional convective systems augmented by rows of film cooling are adequate
Between 1600 K and 2000 K	Advanced (conventional and impingement) convective systems are required which must be augmented by extensive use of film cooling
Above 2000 K	Some form of transpiration cooling with cooled cooling air will be necessary

The remarkable increase in the turbine entry temperature has been achieved by employing mainly three modes of cooling i.e. convection cooling, impingement cooling and (or) film cooling. Bayley and Turner (2) mention nine possible modes of blade cooling which in order of cooling efficiency are as follows

- (1) Simple internal convection cooling
- (2) Impingement (i.e. enhanced convection) cooling
- (3) Film cooling by localised injection through slots or holes

- (4) Distributed injection through drilled sheet
- (5) Distributed injection through coarse wire structures
- (6) Distributed injection through multiply drilled sheets
- (7) Distributed injection through fine wire structure
- (8) Distributed injection through uniformly porous materials, manufactured by techniques of powder metallurgy.
- (9) Distributed injection through porous materials with two-dimensional (e.g. chordwise and spanwise) variations of porosity and hence controlled distribution of cooling air-flow.

Fullagar's (1) design picture as well as Bayley's and Turner's (2) list of cooling methods can be put in perspective by Helms and Emmerson's study (3), from which figure 2 is taken. It is seen from this figure that of all the cooling systems, transpiration cooling provides the most effective mean of cooling turbine blades. However, it also presents some formidable practical problems, mainly because the cooling air contains dust and oil particles (the cooling air is usually by passed the combustion chamber) which block the pores of the transpired blades; consequently the cooling efficiency of the system gradually decreases with the blockage of the pores. Also the transpiring materials (usually sintered wire composites) have low strength at high temperature, low modulus of elasticity and poor oxidation resistance, characteristics which are not desirable for the construction of stator or rotor blades. For the above reasons transpiration cooling, despite its superiority as a cooling method over all the other methods, has not been widely applied in practical situations.

The method of film cooling in which the coolant is ejected along the surface of the blade into the boundary layer of the flow may be

considered to produce an insulating layer (film) between the wall to be protected and the hot gas stream flowing over it. The method of ejecting the coolant through slots arranged on the surface of the blade was applied as early as 1944. However, this method too has the practical disadvantage that it reduces considerably the mechanical strength of the blade and hence of the blade life. This is the reason why slot cooling has not been applied on blades but it has been extensively applied in other situations where there is not a severe strain loading on the component to be protected (e.g. combustion chambers).

For the last ten years attention has been given to a different mode of film cooling: that from discrete holes in which the cooling air is ejected into the flow over the blade through discrete holes arranged on the surface of the blade. This mode of cooling is inferior both to transpiration cooling and to film cooling through slots as far as cooling efficiency is concerned but it possesses the practical advantage (over slot cooling) of retaining in large measure the mechanical strength of the blade. The efficiency of this mode of cooling, however, is more dependent on the cooling geometry (number of holes, arrangement and shape of holes etc) than for slot cooling.

Some formidable practical problems were encountered in the early stages of development of this system in drilling the holes at the proper locations over the blade with the proper size and orientation. A considerable step towards the wide application of discrete-hole cooling method was recently achieved by adopting electrical-discharge machining in "drilling" the discrete holes over the blade. This process removes the blade material by multiple spark discharge action at the end of a special electrode. By controlling the processing parameters, very

accurate, smooth holes can be produced down to 1.5 mm in diameter with low residual stress levels. By changing the shape of the electrode many shapes of holes may be obtained. Figure 3, taken from (3), shows an experimental turbine blade. The holes were drilled by the electrical discharge machining. Three rows of 2 mm diameter holes were produced in the leading edge area while a row of slots $0.25 \times 2.5 \text{ mm}^2$ provide cooling air discharge near the trailing edge on the pressure side of the blade.

With the discrete-hole cooling system there are some associated operational shortcomings. As the turbine entry temperature is increased the quantity of cooling air requires also to be increased; consequently for a given discrete-hole area the injectant velocity of the cooling air is raised considerably. This has as an immediate consequence the reduction of the turbine efficiency due to the aerodynamic losses arising from the mixing process of the cooling air with the mainstream air over the blade. Hence the relation between turbine entry temperature and turbine efficiency has to be constantly in mind as regards the mode and amount of blowing through the discrete-holes.

The discrete-hole cooling system is in operation in many civilian and military aero turbine engines. An example is the Rolls-Royce RB211 engine which powers the Lockheed TRISTAR aircraft. During the development of the RB211 engine, this cooling system had undergone considerable testing and the progress achieved is shown in figure 4, taken from Eltis and Wilde (4). It is particularly noticeable how turbine entry temperatures and thrust have been increased while, at the same time the turbine-blade metal temperatures have been reduced by more effective distribution of cooling air and the application of film cooling especially to the leading edge and

pressure surface of the blade.

From the above discussion one may conclude that film cooling from discrete-holes is the most promising technique for cooling turbine blades. The cooling efficiency of the system however, is seen to depend crucially on the discrete-hole geometry (number of holes, shape and orientation etc) as well as on the operation parameters (amount and distribution of blowing, turbulence level, acceleration parameters etc). Consequently, the present study has aimed to supplement and extend the existing knowledge of discrete-hole cooling systems by studying both experimentally and theoretically the effect of the major parameters on the cooling efficiency of the system. In the following paragraphs the relevant work on the subject is reviewed and the specific tasks undertaken in the present study are stated in more detail.

1.2 A review of previous studies

This review of previous studies considers three main categories of study:

- (i) Jets interacting with a cross-flow.
- (ii) Correlation formulae for film cooling effectiveness and
- (iii) Experimental studies of discrete hole cooling processes.

Attention is mainly given to the experimental studies because, as it will be seen, the analytical methods are primarily applicable to blowing rates greater than unity; they are consequently not of immediate interest to blade cooling, while the correlation formulae, despite the success they have achieved within the limits of their applicability, require extensive empirical information about the flow field in which they are going to be applied.

1.2.1 Interaction of a jet with a cross-flow

(1) Analytical studies

The interaction of a turbulent jet discharging into a stream has been studied analytically by many workers, primarily for blowing rates greater than unity. Platten and Keffer (5) present a mathematical model for the round turbulent jet interacting with a mainstream flow in a wide range of interaction angles and for high injection rates. Integral conservation equations in two dimensions (along the symmetry plane bisecting the jet) for mass and momentum are derived and the extra unknown quantities (apart from the velocity and the inclination of the jet trajectory) are eliminated on the assumption that:

- (a) the shape of the cross-section of the jet remains self similar
(this was found by Keffer and Baines (6) for the normal jet for the region beyond the potential core) and
- (b) the entrainment velocity consists of the entrainment due to ordinary mixing (which occurs because of the difference in relative velocities) and that due to the counter-rotating vortex pair existing downstream of the jet.

The above assumptions introduce into analysis empirical coefficients which are optimised by reference to existing experimental data concerning the jet trajectory. The model proves to be successful in predicting the jet trajectory in the initial region of the developed flow i.e. the section immediately downstream of the potential core; however, further downstream the character of the flow changes as the jet gets absorbed in the cross flow and the model ceases to be successful. Figure 5, taken from (5), shows a comparison between the predicted jet trajectory and the experimental data.

Margason (7) carried out a flow-visualization study to determine the trajectory of a jet directed at large angles to a subsonic free stream. The experimental results of the study were compared with available analytical solutions for the jet trajectories and an assessment for the capability of these methods was made. He presented an empirical equation for the experimental results which correlated the data successfully. He concluded that the main variables determining the jet trajectory from a particular nozzle are the deflection angle of the nozzle exit and the effective velocity ratio.

Chan (8) presented an account of existing theoretical models and concluded that the jet development could not be based on a model applicable throughout the flow field; instead each region of a jet development (initial potential core, the developed region and the far downstream region) had to be treated by involving different assumptions regarding the shape of the jet cross-section, the mode and strength of the entrainment and the importance of the pressure field.

The above theoretical analyses are of interest because they present a fairly accurate description of the jet trajectory for high blowing rates; however, they do not give a clear picture of the rate at which the jet is diluted by the mainstream flow. We could of course argue that the rate at which the jet grows is an indication of its dispersion; nevertheless no attempt has been made to compare the rate of jet dilution with experimental data.

(2) The static pressure distribution around a normal jet to a cross flow

The static pressure field induced by a circular jet issuing normally from a plane wall into an airstream has been studied by Vogler (9), Bradbury and Wood (10) and Chan (8). Vogler (9) carried out an

investigation to determine the pressure distribution on the surface of a flat plate with a jet issuing perpendicularly to the free stream for blowing rates ($1 < M < 5$) that might be encountered in the operation of VTOL aircraft in transition flight. The results indicate that the jet produces a region of positive pressure upstream, a maximum pressure coefficient near the upstream edge of the jet of around $C_p = 0.5$, and a larger region of negative pressures laterally and downstream of the jet. The negative downstream pressure field on the plate is stronger and more extensive than the positive field upstream (C_p is typically about -1.5 at the downstream edge region of the jet). The negative pressures field extend 8 to 10 jet diameters behind the jet and its strength increases with the blowing rate. The pressure field produced on the plate by the jet and that produced by replacing the jet with a rod shows close similarity upstream of the jet but differences downstream partly because the jet is bent by the mainstream and mainstream fluid is entrained into it.

Bradbury and Wood (10) investigated the wall static pressure distribution in connexion with the lift losses observed in models of direct jet lift aircraft. Their conclusions were similar to Vogler's (9); a comparison between their measurements and predictions based on a model of flow around a cylinder with distributed suction around the periphery to simulate the entrainment into the jet was not particularly successful.

Chan (8) carried out a similar investigation for blowing rates $M > 1$ and found that the pressure coefficient C_p falls to the value of $C_p = -0.1$ within 3 jet diameters downstream of the jet. For high blowing rates he confirmed Vogler's (9) finding that the pressure field around the upstream periphery of the jet was similar to that

around the periphery of a cylinder in a cross-flow. Figure 6, taken from (8), shows the static pressure distribution for a blowing rate, M of 2.26

1.2.2 Correlation formulae for the effectiveness of discrete hole cooling systems

For two-dimensional cooling systems (injection through slots or through porous surfaces) a number of formulae have been proposed for correlating the experimental data. These formulae are results either of semi empirical analyses or fitting curves to experimental data. The theoretical formulae are based on energy balance within the boundary layer, the best known of these being quoted by Goldstein* (11). In addition there are some empirical correlation formulae which have been proven successful in a number of cases, for example those of Wieghardt (12) and Spalding (13) based on 2D flow field analyses.

For discrete-hole-cooling geometries, however, the above formulae are not applicable. They usually overpredict the wall film cooling effectiveness, because the character of the flow is now three - rather than two-dimensional. Some studies have reported correlations for discrete hole cooling geometries which achieve some success mainly for low blowing rates ($M < 0.2$ for which the jet stays within the boundary layer and in touch with the surface) provided the holes are not too close together ($S/D > 3.0$, so that the individual jets emerging from each injection hole do not strongly interact with each other). Almost all the derived correlation formulae are based on a theoretical model proposed by Eckert (14). The basic approach of Eckert's model is described in (15); the injection through the discrete hole is simulated

*Reference (11) provides a survey on film cooling developments and it should be consulted for a complete introduction to the field.

by a point heat source moving with constant velocity on a straight line along an adiabatic surface. The dilution of the coolant at any distance x behind the source is given by:

$$\frac{T(x,y,z)}{T(x,0,z)} = \exp \left[- \frac{Pe}{4} \left[\left(\frac{y}{x} \right)^2 + \left(\frac{z}{x} \right)^2 \right] \right] \quad (1.1)$$

where the Peclet number is defined as $Pe = \frac{ux}{a}$ and a is the effective thermal diffusivity. The temperature difference on the plate surface is given by

$$T(x,0,z) = \frac{\dot{q}}{2\pi\rho_c C_p x} \quad (1.2)$$

where

$$\dot{q} = C_p \dot{m} (T_c - T_\infty)$$

and $\dot{m}$ is the mass injected, ρ_c the density of the injected fluid and C_p its specific heat.

Obviously $T(x,0,z)$ represents the wall film cooling effectiveness. The model needs empirical information for the effective thermal diffusivity a . Eckert (14) chose a to fit experimental data at some point in the flow. Once the value of diffusivity was chosen, the model predicted successfully the downstream variation of the film cooling effectiveness for low blowing rates. Figure 7, taken from (14) illustrates the capabilities of this simple model.

Mayle and Camerate (16) applied a generalisation of Eckert's (14) point source model suggested by Eriksen et al (17), to correlate their experimental data^a for single row and multi-row hole arrangements for spanwise averaged film cooling effectiveness. The model of (17) gives the following relation for the adiabatic wall cooling effectiveness.

$$\eta(x,z) = \frac{1}{8} \frac{DMPe}{\sqrt{x^2+z^2}} \exp \left[- \frac{1}{2} \frac{Pe}{D} \left[\sqrt{x^2+z^2} - x \right] \right] \quad (1.4)$$

which is equivalent to (11) for small values of z/x ($z/x \ll 1$). The variation of average film cooling effectiveness with downstream distance was found by determining the spanwise-averaged film-cooling effectiveness for all the coolant jets in a single row and then superimposing the results for each row. For a single row of holes Mayle's and Camerate's results were well approximated by the relation

$$\bar{\eta} = \frac{\sqrt{\pi}}{4} \left(\frac{D}{S} \right) M (Pe/(x/D))^{1/2} \quad (1.5)$$

which states that the average film cooling effectiveness downstream of a single row of holes is proportional to the fractional coverage, D/S , and the mass flux ratio. The streamwise variation of the mean film effectiveness was found by superimposing the effectiveness of the individual rows in the multihole pattern. A linear superposition of the average effectiveness was tried firstly but was unsuccessful. The streamwise variation was then found by using the "additive film" method of superposition which says that the gas temperature used to determine the film effectiveness downstream of an injection location should be the adiabatic film temperature due to all injections upstream of that locations. The final expression for a multirow hole coverage for the spanwise averaged film cooling effectiveness is

$$\bar{\eta} = 1 - \exp \left[- \frac{\sqrt{\pi}}{4} \left(\frac{D}{S} \right) M \sqrt{Pe} \sum_{k=0}^{v-1} (x/D - kS_x/D)^{-0.5} \right] \quad (1.6)$$

where S_x is the streamwise hole pitch and v the number of rows.

Figure 8, taken from (16), shows the performance of the derived correlations. Pedersen (18) and Kadotani (19) introduced into the model of (17) a directional heat transport effect by assuming that the thermal diffusivities in the y and z directions were not equal. The

refined model as applied in (19) had the form

$$\eta(x,z) = \frac{u_{\infty}^2 D^2}{4a_y x} M \exp \left[-\frac{u_{\infty}^2 D^2}{4a_y x} \left(\frac{y_0}{D} \right)^2 \right] \exp \left[-\frac{u_{\infty}^2 D^2}{4a_z x} \left(\frac{z}{D} \right)^2 \right] \quad (1.7)$$

Obviously additional empirical information is required to apply the above relation since apart from d_y , the diffusivity in y direction, values are also needed for a_z and y_0 , the latter denoting some effective height above the surface of the point source.

By choosing these unknowns appropriately Kadotani (19) found that equation (1.7) could correlate his experimental data successfully.

1.2.3 Experimental studies of discrete-hole cooling phenomena

Before we proceed to show the importance and the effect of the various parameters on the cooling performance we shall briefly present the experimental methods which different workers applied in their research.

(1) Experimental Techniques

Two techniques were mainly used to measure the wall film cooling effectiveness. The work at the University of Minnesota, Mech. Eng. Dept. has used a heated secondary jet with thermocouples arranged on the surface of the plate, for measuring the wall temperature and from which the film-cooling effectiveness could be deduced. The same technique was also applied by Liess (20), Liess and Carnel (21), Mayle and Camarate (16) among others.

The second technique, based on the Mass Transfer analogy, was applied by Launder and York (22), Le Brocq et al (23) and, Pedersen (18). The technique consists in mixing with the secondary jet a tracer gas, (studies (22) and (23) used 100% mixture), usually He, and measuring downstream of the jet the concentration of the tracer gas in the flow. From these concentration values an equivalent film cooling

effectiveness can be deduced.

Smith et al (24) applied a short duration-shock tunnel, used as injectant nitrogen and measured by thermocouples the heat flux with and without injection. The ratio of the two fluxes can again be related to adiabatic film cooling effectiveness.

The study of the effect of the width of the boundary layer on film cooling was accomplished by Liess (20); the effective origin of the boundary layer was varied by employing different lengths of test plate upstream of injection.

The effect of streamwise pressure gradient was usually accomplished by varying the position of the top wall of the wind tunnel, Launder and York (22) and Jabbari (25) among others.

(2) Experimental Studies

Before we proceed in the critical review of the experimental studies on discrete-hole cooling systems we present them, in summary, on Table 1 with indication of the parameters examined in each study and the prevailed experimental conditions.

In this review the parameters which affect the performance of the discrete-hole cooling systems have been categorized in two; as operational ($M, \alpha, dp/dx, Tu$ etc) and as geometrical (S/D , shape of hole, etc.). The effect of each parameter on the performance of the cooling system is presented as it is revealed from the various experimental studies. This mode of presentation has the benefit of making clear the importance of each parameter on the efficiency of the cooling system while in the same time it discloses the areas which need further investigation.

Reference	Geometry	α	M	ρ_c/ρ_∞	δ_1/D	$\frac{u_\infty D}{v}$	s/D	$\frac{\nu}{u_\infty} \frac{du_\infty}{dx}$	Tu	Comments
(16)	Multiple row	30°	0.5 1.0 1.5 2.0	1.1	0.32	3600	8,10, 14 " " "	0.0	-	45° lateral injection $\bar{\eta}$
(18)	Row of holes	35°	0.21 0.40 0.52 0.76 1.05 1.50 1.96	.5-4.2 .5-4.2 .75-4.2 .76-4.2 .74-4.2 .75-4.2 .74-4.2	0.156 0.163	19800 11000	3.0	0.0	0.4% 0.3%	η_c
(19)	Row of holes	35°	0.2 0.35 0.50 1.00 1.50 0.2 0.35 0.50 1.00 1.50	0.85	0.245 0.207 0.175 0.19	11000 22000 44000 22000	3.0 3.0	0.0 0.0	0.4% 0.06 up to 0.30	η
(20)	Row of holes	35°	0.16 0.34 0.91 1.43 1.98 0.31-0.43 0.58 1.06 1.53 0.52 1.10 1.54	0.79 0.79	0.171 0.05-0.6 -	- -	3.0 3.0	0.0 0.0	- 75mm Hg cm 150mm Hg cm	$\bar{\eta}$ Mach number 0.3-0.9 The diameter of the injection hole as well as the working pressure are not given
(21)	Row of holes	35°	0.85 0.85 0.85 1.11 0.65	0.71	-	-	2.22 3.33 4.00 2.22 2.22	0.0	-	Mach number 0.4-0.6
(22)	Multiple row	45°	0.1-0.65 0.1-0.65 0.1-0.65	1.5 1.5 1.5	6.4 6.4 6.4	3200 3200 3200	8 8 8	0.0 210 ⁻⁶ 210 ⁻⁶	0.5% 0.5% 6%	Staggered geometry $S_{x/D} = 8$

Table 1 : Summary of experimental studies.

Reference	Geometry	α	M	ρ_c/ρ_∞	$\delta_{1/D}$	$\frac{u_\infty D}{v}$	s/D	$\frac{\nu}{u_\infty} \frac{du_\infty}{dx}$	Tu	Comments
(23)	Multiple row	90° 45° 45° 45°	0.54 0.54 0.54 up to 1.0	1.0 1.0 4.25 1.0-4.25	6.4	2150	8	0.0	0.3%	Staggered geometry $S_x/D = 8$
(24)	Row of holes	90° 30°	0.5-6.0	-	-	1.510^3 1.510^5	2.5	0.0		$\bar{\eta}$
	Double	90° 30°	0.5-6.0				2.5/ 2.2			
(25)	Double row	35°	0.2 0.35 0.50 0.65 1.0	0.84	.177	2.110^4	3.0	0.0	0.4%	$S_x/D = 2.6$
			0.35 0.50 0.65 1.00 1.50		0.231	10^4				
			0.35 0.50 0.65 1.00 1.50		0.196	1.110^4		0.4510^{-6}		
	Double row		0.35 0.50 0.65 1.00 1.50	0.84	0.171	10^4	3.0	1.0510^{-6}	0.5%	
(26)	Single hole	35°	0.1 0.5 1.0	0.84	0.05	0.87_{10}^5		0.0	0.5%	η
			1.5 2.0		0.057	0.44_{10}^5				
(27)	Single hole	35°	0.5 1.0 2.0	0.84		0.44_{10}^5		0.0	0.5%	η_{cl}
			0.5 1.0 2.0			0.22_{10}^5				
	Row of holes		0.5 1.0 1.5 2.0		0.124	0.22_{10}^5	3.0			
			0.5 1.0		0.124	0.22_{10}^5	3.0			15° lateral injection

Reference	Geometry	α	M	ρ_c/ρ_∞	$\delta_{1/D}$	$\frac{u_\infty D}{v}$	s/D	$\frac{\nu}{u_\infty} \frac{2}{D} \frac{du_\infty}{dx}$	Tu	Comments	
(28)	Single hole	35°	0.5	0.85	0.124	0.22×10^5		0.0	0.5%	η	
			0.75								
			1.0								
			1.5								
			2.0								
			0.5		0.116	0.44×10^5					
			0.75								
			1.0								
			1.5								
			2.0								
	Row of holes	0.1	0.114	0.44×10^5	3.0						
		0.2									
		0.5	0.124	0.22×10^5							
		1.0									
		1.5									
		2.0									
(29)	Single hole	90°	0.1	0.85	0.058	0.83×10^5		0.0	0.5%	η	
			0.2								
			0.5								
			1.0								
			0.2		0.067	0.45×10^5					
			0.5								
1.0											
1.5											
2.0											
Row of holes	0.2	0.149	0.44×10^5	3.0							
	0.5										
	1.0										
(30)	Row of holes	35°	0.52	0.84	0.159	1.77×10^4	3.0	0.0			η_L round holes shaped holes
			1.04		0.158	1.83×10^4					
			2.20		0.183	0.87×10^4					
			0.53		0.158	1.73×10^4					
			1.01		0.151	1.91×10^4					
			2.04		0.180	0.91×10^4					
			(31)		Single hole	90°				0.1	
0.5	0.038	0.44×10^5									
1.0											
2.0											
1.0				0.06			0.44×10^5				
2.0											

Reference	Geometry	α	M	ρ_c/ρ_∞	δ_1/D	$\frac{u_\infty D}{v}$	s/D	$\frac{\nu}{u_\infty} \frac{du_\infty}{dx}$	Tu	Comments
(32)	Single row Double row	30° 90° 30°	0.37 } 0.66 } 1.66 } 1.84 } 2.93 } 0.17 } 1.45 } 2.28 } 0.924 } 1.82 }	1.3	0.25-0.5	-	2.5 5.0 2.5	0.0	-	$\bar{\eta}$
(33)	Double row	90°	0.46 1.75 2.59	1.3	0.2-0.5		2.5	0.0		$\bar{\eta}$
(34)	Multiple	90°	0.1 } 0.2 } 0.1 } 0.2 }	~1.0	0.27	3200	4.8	0.0	2%	$\bar{\eta}$ staggered geometry } in line geometry

Operational parameters

(a) Effect of blowing rate on wall film cooling effectiveness

The effect of blowing rate on the wall film cooling effectiveness has been studied by many research workers (16), (18), (19), (20), (21), (22), (24), (25), (26), (27), (28), (29), (30), (31) and (32) for various angles of injection $\alpha = 90^\circ$, 45° and 30° . The major conclusion from all studies is that the wall effectiveness is strongly dependent on the blowing rate. For a single hole and a row of holes the wall effectiveness increases with the blowing rate up to injection ratios around M_{opt} of 0.5 (the value of M_{opt} depends on the particular δ/D , α , $\frac{dp}{dx}$) and then decreases for higher blowing rates. The increase of wall effectiveness for blowing rates $M \leq M_{opt}$ is attributed to the enthalpy decrease of the hot mainstream gas due to the mixing process with a progressively greater amount of coolant injected into the boundary layer; at such low injection rates the secondary jet is bent over by the mainstream flow and remains in touch with the surface to be protected.

For values of M higher than M_{opt} however, the secondary jet, having significant vertical momentum, lifts off the surface and penetrates into the mainstream flow. For a given injection ratio the film cooling effectiveness decreases monotonically with increasing axial distance for lateral positions on or near the centre line of the jet. At sufficiently large lateral distances the effectiveness is seen to increase rather than decrease with increasing axial distance. These positions are beyond the edge of the incoming jet, therefore, at small values of x/D the jet has not spread to those lateral positions. As the jet progresses downstream it spreads laterally, thereby causing the effectiveness to rise. If we take 5% wall-effectiveness contour as representing the effective lateral spread of the jet emerging from a

single hole, one may conclude that for 90° injection the jet does not spread laterally further than 1.75 hole diameters and for 35° injection not greater than 1.30 hole diameters ((26),(27)). For the latter case it is obvious that for a row of holes, a hole spacing greater than 2.6 diameters would yield effectiveness results almost identical with those for a single hole under the same experimental conditions, since the individual jets emerging from the holes do not interact with each other for at least 20 hole diameters downstream of the injection hole, Goldstein et al (27).

Velocity measurements taken by Ramsey and Goldstein (31) for 90° , single-hole injection show that, for $M=1.0$, there exists a region of low velocity behind the hole near the wall; above this region the velocity increases reaching a maximum which is greater than the free-stream velocity and then decreases to the free-stream value. In fact flow visualisation studies by Ramsey and Goldstein (31) show that for normal injection there is a region of reverse flow behind the hole. The hot wire anemometry technique used in the same study does not distinguish backward-from forward-moving fluid and consequently their measurements behind the injection hole show the resultant of the (weak) reverse flow and the vertical velocity to be high approximately 30% of the free-stream velocity

(b) Effect of angle of injection

A second parameter significantly affecting the wall effectiveness is the angle of injection to the mainstream flow. Most of the studies have been for injection angles of 90° or 35° and a few for 45° . The general conclusions of the experimental studies for normal and inclined injection are that for the same blowing rate there is less penetration into the external mainstream (i.e. less vertical and lateral spread) for

the angled injection. The reason for the smaller vertical spread lies in the fact that the jet, emerging from an angled hole, has smaller vertical momentum than if it had been issued from a normal hole. The lateral spread of the jet is mainly controlled by diffusion process. Normal injection creates greater mixing than inclined injection because it produces greater mean rates of strain at the edge of the jet and consequently greater effective diffusion coefficients. The following example helps quantify the position. Measurements for 90° and 35° injection for an injection ratio of $M=1.0$ by Ramsey and Goldstein (31) show that for angled injection the 5% effectiveness at location $x/D=12.0$ and $z/D=0.0$ lies 1.8 hole diameters above the wall compared with 2.5 hole diameters for normal injection. However the lateral spread for the inclined injection is much smaller, the 5% wall effectiveness at $x/D=12.0$ being at 0.75 hole diameters laterally compared with 1.5 hole diameters for normal injection.

Comparison of measurements of Eriksen (28) for injection ratio $M=0.2$ for normal injection (single hole) and 35° (row of holes with $s/D=3.0$) shows that at a downstream location of $x/D=12.0$ the centre-line effectiveness for inclined injection is approximately 15% higher than for normal injection, while the 5% wall effectiveness is at lateral location 1.4 hole diameters for inclined injection in contrast to 1.8 hole diameters for normal injection. Figures 9(a,b), taken from Eriksen's study (29), reinforce the above arguments.

For a "full coverage" cooling geometry LeBracq et al (23) studied the effect of varying the angle of injection (90° and 45°) on cooling effectiveness. The conclusions drawn were in line with the results mentioned above: namely, that the advantage, arising from inclined injection, of higher effectiveness near the centre line is at least

partly offset by the smaller lateral spread.

Goldstein et al (28) compared the laterally averaged film cooling effectiveness for normal and inclined injection and concluded that the mean effectiveness is higher for inclined injection (35°) for blowing rates for which the jet stays in touch with the surface ($M \leq 0.5$); for an injection ratio of $M=1.0$, however, where in both cases the jet lifts off the surface, normal injection gives somewhat higher mean effectiveness than the inclined one. The overall level of mean effectiveness for $M=1.0$ is lower than the corresponding one for $M=0.5$, however.

(c) Effect of boundary layer thickness

The wall film cooling effectiveness decreases as width of the boundary layer increases. An explanation for this behaviour lies in the fact that the thinner the boundary layer, the higher the velocity of the near-wall mainstream fluid and consequently the greater the force acting on the jet emerging from the hole; the jet is thus forced to stay in touch with the wall resulting in higher wall effectiveness.

Quantitatively the effect has been studied in (19), (20), (28), (29), (30), (32). In the work of University of Minnesota group (19), (28), (29), (30), the study covered the range $0.05 < \delta_1/D < 0.25$. Over this range of displacement thickness and for angled injection of 35° they reported that for all the injection ratios ($0.1 < M < 2.5$) except $M=0.5$, the boundary layer thickness had only a small effect on the effectiveness (5-10%), the effect being smaller at larger δ_1/D , while for $M=0.5$ the boundary layer thickness did have significant effect. In particular they found that by doubling the displacement thickness the centre line effectiveness was reduced by 30% for $M=0.5$.

For 90° injection the boundary layer thickness was found not to be a significant parameter.

Liess (20) studied the effect of boundary layer thickness covering a much wider range of displacement thicknesses ($0.05 < \delta_1/D < 0.65$) than in the University of Minnesota study. He concluded that for a displacement thickness $\delta_1/D < 0.2$ and for the studied injection ratios $M = 0.3-0.4$ ($\alpha = 35^\circ$), the boundary layer thickness had small influence on film cooling effectiveness (in line with the conclusions of the Minnesota group). For $\delta_1/D > 0.2$ he found that the effectiveness decreased considerably with increasing boundary layer thickness. In particular for a location $x/D = 12$ for $\delta_1/D = 0.1$ the centre-line effectiveness was measured as $\eta_c = 0.23$ while at the same location for $\delta_1/D = 0.65$ the effectiveness was $\eta_c = 0.10$ a decrease of more than 50%. He concluded that the displacement thickness is of more importance than the shape of the velocity profile itself. This effect can be plainly seen on figure 10, taken from (20).

Kadotani (19) studied the effect of the boundary layer thickness on the off centre-line and laterally averaged cooling effectiveness as well. He concluded that the effect of boundary layer thickness is pronounced on the region near the injection hole and then diminishes with distance downstream. The effectiveness at lateral locations increases with increasing boundary layer thickness because the jet penetrates more into the mainstream and consequently higher interaction results; thus as discussed above with reference to normal injection higher effective transport coefficients are produced. The laterally averaged effectiveness was found to increase slightly (5%) as the boundary layer thickness was reduced, the smaller η_c for greater δ_1/D being nearly offset by the greater lateral spread.

(d) Effect of density ratio of coolant to mainstream

In practical situations the coolant, being at a lower temperature than the mainstream, will be denser than the mainstream by a factor of about 1.5. The simulation of the denser secondary jet in laboratory experiments has usually been achieved by using carbon dioxide ($\rho_c/\rho_\infty=1.5$) or Freon ($\rho_c/\rho_\infty=4.0$) or mixtures of these gases with air for different density ratios. An extensive study on wall effectiveness of this particular parameter has been made by Pedersen (18) as well as by Goldstein et al (30) and Le Brocq et al (23)

The results suggest that the denser coolant gives a higher level of film cooling effectiveness mainly for injection ratios $M \geq 0.5$ (for which region the jet lifts off the surface and penetrates into the mainstream). The reason for this behaviour is that the coolant being denser has smaller injection velocity for the same mass flux, thus smaller vertical momentum; it is thus more easily bent by the mainstream flow and remains closer to the surface. For injection ratios lower than $M=0.5$ however, where the jet (for $\rho_c/\rho_\infty=1.0$) would remain attached even when there was no appreciable density difference there is no significant effect of ρ_c/ρ_∞ on the film cooling effectiveness.

Pedersen (18) also reports mean spanwise effectiveness as a function of the density ratio from which study figure 11 is taken. The results suggest that by increasing the ratio of coolant to mainstream densities the mean effectiveness increases substantially mainly for blowing rates greater than $M=0.5$. For $\rho_c/\rho_\infty=1.5$ (close to that which would occur in practice) the results show that the optimum injection ratio (i.e. that giving maximum spanwise averaged effectiveness) at $x/D=12$ is around $M=0.5$ while at $x/D=41$ is $M=1.0$. However, for the latter case the curve is flat ($\bar{nv} \sim M$), and the maximum is barely 10% more than obtained

for $M=0.5$. Thus one may conclude that for $\rho_c/\rho_\infty=1.5$ and $S/D=3.0$, $M=0.5$ is an optimum value for this cooling geometry, the same as with $\rho_c/\rho_\infty=1.0$.

(e) Effect of lateral injection

It was mentioned earlier that by reducing the angle of injection the central region effectiveness near the centre is increased but the lateral spread of the jet is decreased thus requiring smaller hole spacing for an adequate protection of the surface. However the addition of more holes adversely affects the strength of the blade and increases its cost. One way of increasing the lateral spread of the jet is to inject the coolant laterally at an angle to the mainstream flow.

Goldstein et al (27) studied a case where the coolant was injected 35° to the surface making a zero angle with the streamwise direction and 15° and 35° to the surface making an angle of 90° with the mainstream. It was found that for an injection ratio of $M=0.5$ for which the jet stays on the surface the lateral injection widened the protected area but reduced the maximum effectiveness. The reason for this behaviour can probably be attributed to the stronger interaction between the jet and the mainstream, due to the lateral injection.

For injection ratio $M=1.0$, for which the jet penetrates into the mainstream flow, the lateral injection results both in wider region of protection and higher effectiveness, meaning that the lateral injection holds the jet closer to the wall and is more effective from the film cooling point of view.

(f) Effect of streamwise pressure gradient

In practical situations there will be favourable streamwise pressure gradient over the blade. The effect of pressure gradient on film cooling

effectiveness has been studied by many workers, (19), (20), (22) and (25).

Launder and York (22) reported that a moderate acceleration parameter ($\frac{v}{u_\infty^2} \frac{du_\infty}{dx} = 2.10^{-6}$) produces substantially higher film cooling effectiveness than a zero pressure gradient while Kadotani (19) concludes that an acceleration parameter up to 10^{-6} reduces averaged film cooling effectiveness by approx. 5%, mainly well downstream from the injection holes. Liess (20) has likewise reported that a favourable pressure gradient reduces slightly the film cooling effectiveness. Jabbari (25) studied the effect of streamwise pressure gradient on effectiveness for a double row of staggered holes ($s/D=3.0$, $Sx/D=2.6$) with a turbulent secondary jet. His conclusions were that for the range of acceleration parameters studied $\frac{v}{u_\infty^2} \frac{du_\infty}{dx} = 0-10^{-6}$, an accelerating flow causes only a slight reduction in effectiveness along the centre-line downstream of the hole. The same effect is also noticed on laterally averaged effectiveness.

The difference in the effect of streamwise pressure gradients in the studies of (22) and those of (19), (20) and (25) lies in the fact that in the former study the secondary jet is laminar while in all the other cases it is turbulent. Thus in Launder and York's work (22) the reason for the higher film effectiveness is that the acceleration tends to diminish turbulent mixing in the external boundary layer and offers a stabilizing environment to the laminar jet emerging from the hole. In these conditions the jet undergoes transition to turbulent flow much less rapidly than it does in parallel flow; the streamwise dilution rate is thus reduced and the lateral spread is decreased.

In the studies of Liess (20), Kadotani (19) and Jabbari (25) the secondary jet is turbulent and one expects that a favourable pressure

gradient would result in higher effectiveness by reducing the turbulent mixing between the secondary jet and the mainstream. Liess (20) reports that the reduction of the film cooling effectiveness is associated with the suppression of the streamwise vortices behind the injection hole by the favourable pressure gradient. (A possible explanation for this behaviour is that the favourable pressure gradient suppresses the upward movement of the vortices and forces them to stay nearer to the surface and consequently to augment the dilution). Of course the same mechanism is present when the secondary jet is laminar but it seems the mechanism of delaying the transition of laminar jet to turbulent is more effective than the action of the vortices.

(g) Effect of free stream turbulence level and turbulence length-scale

Launders and York (22) and Kadotani (19) have studied the effect of turbulence level on wall film-cooling effectiveness. Launders and York (22) draw the conclusion that the presence of free-stream turbulence intensities of about 4 per cent reduce the wall film cooling effectiveness by typically 10 per cent in the case of a favourable pressure gradient but have negligible influence for parallel flows. The above behaviour is related to the fact that the secondary jet of the study was laminar and the behaviour is not similar if the jet were turbulent.

Kadotani (19) using a turbulent secondary jet studied the effect of a free-stream turbulence (0.3% - 20.6%) and free stream turbulence scale (0.06 - 0.33 hole diameters) on cooling effectiveness. The effect of free-stream turbulence scale on the wall effectiveness is very large. The author concluded that when the length scale of turbulence is small the effectiveness is high and wall effectiveness contours stretch in the downstream direction. When the length scale of turbulence is large the contours do not stretch downstream but spread more laterally (for

quantitative details see (19)). The effect of the free-stream turbulence depends on the blowing rate. For small blowing rates ($M=0.2-0.35$) the turbulence intensity is very influential on the wall effectiveness. Near to the injection ($x/D < 7.0$) for a blowing rate of $M=0.2$ it was found that a change of Tu from 2.7% to 20.6% brought a 27% reduction on the centre-line effectiveness. The effect on the laterally averaged effectiveness is similar but weaker. For medium blowing rates ($M=0.5$) the centre-line effectiveness slightly increases for large scale turbulence and decreases for small scale turbulence with increase of turbulence level. For large blowing rates ($M > 1.0$) the injected fluid generates very high turbulence and effectiveness while becoming relatively insensitive to the turbulence level, actually increases with increasing turbulence level.

Generally from the study it may be concluded that the turbulence scale has a greater effect on the effectiveness than the turbulence intensity due to the direct effect of the length scale on the magnitude and strength of the vortex motion.

(h) Effect of Mach number

The effect of the free-stream Mach number on the cooling effectiveness is difficult to estimate (independently of the other parameters) because by varying the free-stream velocity, the thickness of the boundary layer at the point of injection is changed and consequently the effect of u_{∞} on η can not be isolated from the effect of δ on η . However, Liess (20) reported a study of the effect of u_{∞} on η keeping the thickness constant. This was accomplished by varying the effective origin of the boundary layer. For the range of Mach numbers $Ma = 0.3-0.9$ and for mass flux ratios $M = 0.4$ he reported that u_{∞} has no effect on the wall effectiveness

(discrepancies within the experimental uncertainty 1%). Goldstein

et al (28) and Eriksen (29) reported similar conclusions based on the study of the combined effects of δ and u_{∞} on film cooling effectiveness. This conclusion is of particular interest since it shows that laboratory experimental results for low free stream velocities are applicable to high free stream velocities occurring in most practical situations.

(i) Effect of the secondary jet velocity profile

Goldstein et al (30) studied the effect on film cooling effectiveness of the secondary jet profile. In particular two arrangements were tried. In the first one the secondary air was supplied to the wind tunnel through a long circular pipe so a well established turbulent pipe flow velocity profile would prevail at the exit of the injection hole while in the second arrangement the length of the pipe was very short so the injected fluid would have a uniform velocity profile. The study concluded that there was no influence on film cooling effectiveness of the secondary profile.

Geometrical parameters

(j) Effect of hole spacing

The effect of hole spacing on the film cooling effectiveness was studied by Goldstein et al (30) and Liess and Carnel (21).

Goldstein et al (30) examined how well the "shaped" hole (round hole widening 10°) would function for hole spacings greater than $3D$. The holes were spaced 6 diameters apart and the results show that the width of the film-cooled region downstream of the holes was not significantly greater than for $3D$ hole spacing, indicating that if such a wide spacing were used there would be a large region between the holes which would not be directly protected by the coolant.

Liess and Carnel (21) studied three cooling arrangements, those of $s/D=2.2$, $s/D=3.3$ and $s/d=4.0$ results of which study are presented in figure 12 for injection ratios $M=0.65$, 0.85 and $M=1.1$. The conclusions are that by increasing the hole spacing the film-cooling effectiveness decreases considerably (especially for $s/D=2.2$ to $s/D=3.3$) for the high blowing rates studied, suggesting that there is a strong interaction between the jets. For a hole spacing ($s/D=2.2$) the flow is almost 2-dimensional beyond 13 diameters behind the hole at least for the wall adiabatic effectiveness .

(k) Effect of hole geometry

Goldstein et al (30) have studied the effect on film cooling effectiveness of the geometry of the injection hole. Experiments were carried out for a round injection hole and a "shaped" hole (a hole which had been widened at each side by an angle of 10 degrees). The results suggest that the "shaped hole" gives higher film cooling effectiveness and higher averaged effectiveness too. The higher lateral cooling coverage is a consequence of the lateral widening of the injection hole, while the greater effective area of the injection hole reduces the injection velocity (for the same amount of coolant) and causes the jet to remain closer to the surface (for $M > 0.5$) resulting in higher levels of effectiveness.

(l) Effect of a slit located behind the injection hole

In the study by Krusse (32) a slit was located at the downstream edge of the injection hole through which 5% of the total secondary flow rate was injected into the boundary layer. The surprising result was that for blowing rates higher than $M=0.5$ the existence of the slit improved the central line region effectiveness considerably, mainly for

blowing rates $M > 1.0$. The effect of the slit reduced as the injection ratio was decreased below $M=0.5$. Preliminary results by Brown (33) confirm the above findings for blowing rates $M > 0.5$.

The reason for this behaviour lies in the lifting off of the jet for blowing rates higher than $M=0.5$, thus the extra coolant from the slit fills in the "stagnant" region (mainly mainstream fluid) behind the hole. Obviously the effect of the slit is reduced for injection ratios lower than $M=0.5$ since the jet then stays attached to the surface.

(m) Effect of multiple rows

Due to the small lateral spread of the jets and to the fact that structural reasons do not permit small hole spacings, it is almost certain that in future multihole coverage (i.e. two or more rows of holes) will be employed.

This cooling geometry has been experimentally studied by many workers (16), (22), (23), (25). The basic finding of all the studies is that for the same amount of coolant injected into the boundary layer the multi hole arrangements are far superior to a single row of holes. There are also aerodynamic advantages with multi-row coverage in comparison with single row for which no assessment has been made at the moment. The main findings can be summarised as follows.

- (i) a staggered pattern of holes is far superior to an in-line arrangement, Le Brocq et al (23)
- (ii) The lateral spacing between holes should be less than eight diameters (about five diameters appears adequate for multiple row coverage (23)) in order to prevent the appearance of "hot" spots.
- (iii) Mayle and Camerate (16) found that for all hole spacings considered and for low to moderate blowing rates, wall effectiveness rises

quite rapidly near the upstream end of the blowing region, but the rate of increase gradually diminishes with streamwise distance as if approaching an equilibrium within the film cooling array.

In particular Jabbari (25) conducted an experimental study for a double row of staggered holes with lateral spacing $S/D=3.0$ and streamwise spacing $S_x/D=2.6$. He concluded that the effectiveness along the centre-line decreases continuously with the distance downstream. The general level of wall effectiveness increases with the blowing rate up to $M=1.0$ and remains almost constant for $M=1.5$. This continuous increase of effectiveness with blowing rate contrasts with the single row data under similar experimental conditions (for which the effectiveness for $M=1.0$ is lower than for $M=0.5$) indicating that there is strong interaction between the jets of the two rows of holes which result in the jets remaining attached to the surface. He also confirmed the finding of Le Brocq et al (23) that injection through a staggered row of holes is superior than an in line row and this superiority is most pronounced at higher values of blowing rates.

Injection through a staggered row of holes over the suction surface of a turbine vane has been investigated by Lander et al (35). Injection angles of 35° and 32° for upstream holes and 45° and 42° for downstream holes were used in this study with a lateral hole spacing of a 2.78 hole diameter. Results indicate that the maximum effectiveness occurs for blowing rates $M=0.5-0.6$ at short distances downstream from the injection point. At locations far downstream however, the effectiveness increases monotonically with increasing blowing rate.

1.3 Outline of the present study

1.3.1 Objectives of the present research program

The previous review of the work on discrete-hold blade-cooling systems disclosed areas which need clarification or further investigation. The studies have all been predominantly experimental and the few analytical attempts (in the form of correlation equations for film-cooling effectiveness) achieve only limited success in view of the empirical inputs required.

To solve the governing differential equations of the flow field in such a complex system has hitherto not been practicable because the flow field is three-dimensional and turbulent.* However, with the application of three-dimensional numerical procedures to some simpler flows (e.g. McGuirk(38), Patankar et al (39)) we felt it would be appropriately challenging to tackle this flow problem. Consequently the first objective of this study was to provide a numerical scheme and a computer program** which was flexible (as regards the initial, boundary conditions and cooling geometry), easy to handle and of course providing accurate predictions which could be used for design purposes.

* For two-dimensional flow fields arising from transpiration cooling theoretical treatments such as those of Baker and Launder(36) and Bayley et al.(37) are available.

** The starting point for the development of the computer code would be the TEACH 3-dimensional fully elliptic program developed by Gosman, Humphrey and Vlachos.

To validate such a numerical procedure there is need for detailed, well documented, experimental data, for both the hydrodynamic and temperature fields. As the review of experimental studies has shown, however, previous experiments have been primarily concerned with measurements of film cooling effectiveness; indeed, apart from some measurements by (20) and (31) few have reported velocity measurements or any other hydrodynamic characteristic of the flow field. Moreover, all workers have focused attention on the region several diameters downstream from the injection hole rather than around the injection hole or over the hole itself. Consequently, the available experimental data are not enough to validate the numerical procedure. In view of these limitations an extensive experimental programme was formulated with the primary purpose of studying experimentally flow situations which could be used as test cases for the elucidation of the numerical procedure. Particular emphasis was placed on:

- . Detailed investigation of the flow field around and far downstream of the injection hole.
- . Detailed study of the effect of boundary layer thickness on the efficiency of the cooling system so as to extend the region of boundary layer thickness studied by the Minnesota group.
- . Detailed investigation of the flow field created by injection through a double row of holes and in particular the region between the two rows in which ref. (32) indicated the existence of a minimum of effectiveness.

1.3.2 Layout of the thesis

The thesis is organised into six chapters.

- . Chapter 1 has introduced the practical relevance of discrete-hole cooling systems, reviewed the available studies on the subject and outlined the objectives of the present research.
- . Chapter 2 describes the apparatus and experimental techniques used in the study and evaluates the accuracy of the measurements.
- . Chapter 3 presents and discusses in detail the experimental results of the study.
- . Chapter 4, which contains the mathematical and physical model of the calculation procedure, is divided into three sections. In the first, the mathematical foundation of the problem and the differential equations governing the flow and temperature field are presented. In section 2 the mathematical modelling of turbulence is discussed, and finally in section 3 the numerical procedure for solving the differential equations is described.
- . Chapter 5 presents the results obtained by the use of the developed numerical procedure and makes comparison with experimental data.
- . Finally, Chapter 6 summarises the achievements of the work, outlines areas of potential use of the prediction method and stresses the areas which need further investigation.

CHAPTER 2

APPARATUS AND MEASUREMENT TECHNIQUES

2.1 Introductory remarks

In this chapter we describe in some detail the apparatus and the measurement techniques employed in the present experimental study, as well as the experimental uncertainty, reproducibility and sensitivity associated with the measurements.

The measurements are such as to give a clear, quantitative picture of the flow field associated with discrete-hole cooling systems, particularly in the vicinity of the injection hole. Consequently, detailed velocity and static pressure measurements are taken in the flow field. Moreover, to reinforce the picture of the jet dilution with the mainstream, a tracer gas (He) is mixed with the jet and its concentration is measured in the flow near the injection hole as well as further downstream.

In table 2 there is a summary of the experimental conditions which prevailed in the present study.

2.2 Apparatus

2.2.1 The Wind Tunnel

The measurements were conducted in the open-circuit wind tunnel shown in figure 13. The mainstream air was drawn from the laboratory via a 15 hp blower and was introduced via a carefully designed diffuser into the settling chamber (equipped with screens) and then via a high contraction ratio nozzle to the test section of the tunnel. The entrance to the test section (figure 14) was specifically designed to bleed off the boundary layer which was created on the bottom wall of the nozzle and thereby supply a uniform-velocity stream of air of nominal velocity 26m/s.

geometry	α	M	δ_1/D	$\frac{\rho}{\rho_\infty}$	S/D	$\frac{U_\infty D}{V}$	$\frac{v}{U_\infty} \frac{du_\infty}{dx}$		Comments
single hole	90°	0.046 0.1 0.24 0.50 1.00	0.048	1.0	-	3.8510 ⁴	0.0	0.4%	Measurements • Pressure • Streamwise velocity • Wall film cooling effectiveness D = 22.22mm
single hole	30°	0.1 0.2 0.3 0.5 1.0 1.5	0.122	1.0	-	3.310 ⁴	0.0	0.4%	Measurements as above D = 19.05mm
single row	30°	0.25 0.50 0.75 1.00 1.50	0.078 0.095	1.0	2.67	3.310 ⁴	0.0	0.4%	Measurements • Pressure • Wall film cooling effectiveness D = 19.05mm
double row	30°	0.25 0.50 0.75 1.00	0.078 0.095	1.0	2.67	3.310 ⁴	0.0	0.4%	Measurements as above • Streamwise hole Pitch $S_{x/D} = 3.50$ D = 19.05mm

TABLE 2: Summary of the Experimental Conditions of the Present Study

The side walls of the test section were constructed from perspex to facilitate flow visualisation. The angle of inclination of the flat roof of the test section was adjustable to allow for adjustment of the streamwise pressure gradient, within limits. For most of the studies a zero streamwise pressure gradient was desired and the roof was therefore adjusted to slightly diverging so as to compensate for the pressure gradient which would otherwise be created by the development of the boundary layers on the side walls of the test section, and by the injection process.

2.2.2 The Secondary Air Supply System

The secondary air for injection was obtained from the laboratory compressed air system whose supply pressure was around 6 atm. The pressure was reduced and controlled by three pressure regulators connected in parallel, as illustrated in figure 15. The air from the supply lines was fed into a plenum chamber of side 0.25m, from which the secondary air was led to the injection pipes. The internal diameter of the injection pipes was $D = 22.22\text{mm}$ for the 90° injection and $D = 19.05\text{mm}$ for the 30° studies. The length of the injection pipes downstream from the orifice plate was at least 50 hole diameters. Each injection pipe had its own needle valve for independent control of the flow rate.

2.2.3 The Test Plates

In the present study three different test plates were used for the following purposes:

- (a) single hole injection normal to the mainstream
- (b) single hole or single row of holes inclined at 30°
- (c) double row of holes inclined at 30°

The test plates were constructed from perspex and around each injection hole were drilled, wherever constructional limitations allowed, static pressure tappings of diameter 0.79mm. These were also used to measure the wall concentration of the tracer gas. Figures 16(a-c) contain photographs of the various test plates, in which the location of the static pressure tappings can be seen. The precise locations will be given later.

A strip of sand paper was fixed to the upstream edge of the test section in order to promote turbulent flow. The dimensions of this strip as well as those of the test sections are given in figures 17(a-d) for the various experimental cases.

2.3 Measurement techniques

Qualitative as well as quantitative information was obtained for the flow field during the present study. The former was acquired by various flow visualisation techniques while the latter employed a variety of measurement techniques.

2.3.1 Flow visualisation techniques

Two methods were used to obtain qualitative information about the flow field. In the first method, the jet emerging from the injection hole was made visible by introducing into the secondary flow a mixture of carbon dioxide and vaporised oil created by a smoke generator and then the jet was photographed. The second technique was aimed at revealing information about the flow field near the plate surface "to be protected", and two different methods were employed. In the first the test-plate surface is coated with a solution of saturn yellow pigment with white spirit and kerosene and is exposed in the test

section for about five minutes with the mainstream and secondary jet turned on. As the plate dries, the great majority of the powder is blown downstream; the residue leaves, on the surface, a clear impression of the streamline pattern in the immediate vicinity of the injection hole.

The second method is applied to reveal local information about the surface flow pattern mainly around the injection hole and it consists of spreading in localised areas of the test plate black lubrication oil and turning the mainstream and secondary flow on. The pattern which is formed as the oil is moving is photographed.

2.3.2 Quantitative measurements

The following quantities were measured in the present study:

- . flow rates of secondary air;
- . distributions of static and dynamic pressure within the flow;
- . surface static pressure;
- . velocity and turbulence intensity at jet entry and
- . surface concentration.

Details of the measurement techniques for these will now be given.

(1) Secondary flow rate measurements

The mass rate to each injection pipe was measured by an orifice plate of nominal diameter $\frac{1}{2}$ " (for blowing ratios $M \geq 0.2$) and $\frac{1}{4}$ " (for $M \leq 0.2$). The orifice meter was constructed according to British Standard BS1042 with D/2 and D tappings. The calibration of the orifice plate was tested by comparing the flow rate calculated from the standard orifice formula and the measured pressure drop with

the value obtained by measuring the outlet velocity distribution for normal injection in the absence of mainstream flow, using a hot wire anemometry. The agreement between the two methods was better than 1%.

(2) Dynamic and static pressure measurements

Measurements of the distribution of static and dynamic pressure within the flow, excluding the region immediately above the injection hole, were obtained with the aid of pitot-static probes of the two sizes shown in figure 18(a-b). When these were employed in regions where appreciable variations in static pressure were observed to occur, the dynamic and static heads were measured separately, at corresponding locations. The probes were aligned with the x direction of the flow (streamwise) and not with the local velocity vector. Consequently, small errors are introduced in the measurements of the velocity which are estimated in the Appendix A. The pitot-static probe of figure 18a was constructed according to NPL standards (40). The smaller one in figure 18b was constructed from thin hypodermic tube and was employed for measuring in off centre-line locations. It was calibrated against the NPL pitot-static probe by measuring the dynamic and static pressure in the wind tunnel without injection; both gave identical results. The pitot-static probes were mounted on a vertical traversing mechanism, shown in figure 18c, which could slide along the test section. The zero vertical positioning of the probes was estimated when the probe coincided with its mirror image on the test plate surface. Repeated estimates of the zero location were in agreement to within about 0.05mm.

The pressures registered by the probes and the pressure tappings were determined by the use of an electronic digital micro-manometer capable of measuring pressure differences from 0.001-2000mm H₂O.

(3) Velocity and turbulence intensity measurements in secondary jet

For these measurements a hot-wire anemometer coupled to an ancillary on-line signal processing equipment was used. This set-up, shown in figure 18d, extensively used by Ribeiro(41), consisted of:

- . the anemometer
- . a lineariser
- . an analogue to digital converter
- . and a PDP computer with its teletype.

Ribeiro(41) gives complete details of this assembly. Briefly the circuit is as follows. The signal from the hot wire was transmitted to the anemometer, to the lineariser and to the digital converter. The digital converter output was fed to the PDP computer. Approximately 10,000 instantaneous values were taken by the computer over a period of 10 seconds and the output from the signal processing unit in the form of mean velocity and turbulence intensity was printed out on the on-line teletype.

The hot-wire probe was calibrated in the potential core of a jet with a nozzle diameter of 25.4mm emerging from a low turbulence (0.2%) wind tunnel specifically designed for calibration purposes. The jet exit velocity could be varied from 3 m/s to 35 m/s. A specially designed gear allowed rotation of the hot-wire probe up to 180°. The values of the velocities calculated from an NPL pitot

tube measuring at the same location the jet with the hot-wire probe were used in determining the constants of the response equation of the probe. Further details of the measurement technique are given in ref. (41).

(4) Concentration measurements

It is known that the adiabatic wall effectiveness is equal to the impervious wall effectiveness for turbulent flows because the Lewis number is unity. The above argument was experimentally verified by Pedersen(18). In the present study this mass-transfer analogy was used to measure the wall effectiveness, because it is easier to satisfy the experimental condition of an impervious to mass than impervious to heat (adiabatic) wall. Consequently, a small quantity of a tracer gas (He) was mixed with the secondary stream (approximately 0.5% by volume) upstream of the plenum chamber. The concentration of helium on the surface of the test plate, from which the effectiveness was deduced, was obtained by withdrawing samples of the air: helium mixture through the static pressure taps and passing them through an on-line conductivity cell. The conductivity cell, which was kept in an oven at 50°, consisted of two chambers containing platinum filaments and which formed the two resistances of a Wheatstone bridge.

The principle of the technique is as follows: the sample and a reference sample (pure air) at the same flow rate (100 ml/min) were introduced into the two chambers of the conductivity cell. The different thermal conductivities of the two samples change unequally the temperatures of the two filaments (hence their resistance) of the cell, thereby creating an imbalance in the Wheatstone bridge. The imbalance of the bridge is related to the amount of

the tracer gas existing in the sample and the effectiveness is obtained as

$$\eta = C_w / C_{inj} \quad (2.1)$$

where C_w and C_{inj} are the concentrations of He at the surface (at the measuring location) and within the injection pipe respectively. The measurement of C_w and C_{inj} are accurate only within limits of the sampling rate and the amount of He mixed with the secondary air. The numbers quoted earlier, that of the sampling rate 100 ml/min and He mixed with air, 0.5% per volume, were determined such that values around these do not have any effect on the measured values of C_w and C_{inj} .

The calibration of the measuring equipment was carried out by previous workers, Rastogi(42) and Baker(43).

2.4 Accuracy of the measurements

An accuracy estimate of the measurements is included in Appendix A. The results are given below.

The pressure difference measurements were estimated to be accurate to 0.2%. However, the accuracy of the velocity profiles deduced from the total and static pressure readings after allowance for the errors associated with yaw and turbulence was estimated to be of the order of 2% in all regions apart from the zone immediately downstream of the injection hole, where, as shown in Appendix A, the disturbances provoked by injection give rise to high errors.

The error in the measurement of the wall static pressure distribution was mainly due to small departures from the nominally

zero streamwise pressure. It is estimated that such departure affects the measured C_p ranging from zero at the upstream row of holes to about 10% of the recorded values at the downstream row and this for the small blowing rate ($M = 0.1$).

The accuracy in the measurements of the velocity and turbulence intensity at the exit of the 90° injection is, according to the error analysis of Ribeiro and Whitelaw(44), in the absence of the mainstream flow, expected to be 1% and 8% respectively. However, the presence of the main flow, with the attendant distortion of the streamlines and generation of high turbulence levels prevented a meaningful estimation of the accuracy of the measurements for those circumstances.

The accuracy of the film cooling effectiveness measurements is estimated to be at best 2% (for $\eta > 0.80$) and at worst 20% (for $\eta < 0.05$).

2.5 Reproducibility of the measurements

Velocity profiles measurements were periodically repeated in order to test the repeatability of the measurements. In all cases the velocity measurements were repeated within 3% even in the region behind the injection hole.

Also periodical tests were performed for checking the thickness and lateral uniformity of the boundary layer on the test plate in the absence of the secondary flow. Samples of film cooling effectiveness measurements were repeated and the statistical spread of most of the measurements was 5%.

2.6 Operating procedures and experimental conditions

2.6.1 Operating procedures

Prior to the final measurements the following operating procedures were performed.

(i) The main and secondary flows were set in operation for approximately 15 minutes.

(ii) The roof of the test section was then adjusted so as to reduce the streamwise pressure gradient to a minimum value.

(iii) The electrical instruments were switched on for sufficient time to reach a steady state condition (the oven of the conductivity cell was switched on one hour prior to the measurements).

2.6.2 Experimental conditions

All the experiments were carried out for a free-stream velocity of about 26 m/s and density ratio of the injectant fluid to the mainstream ρ_c/ρ_∞ , of unit. A summary of the various experimental conditions with the studied parameters is shown in table 2.

The development of the boundary layer velocity profile along the test plate, with the injection holes covered by thin tapes, was recorded and is shown in figure 19a for the case of the single hole 30° injection test plate; the inferred non-dimensional displacement thickness variation is plotted in figure 19b for the same case. The displacement thickness lateral variation was found to be uniform within 0.2% throughout the experimental study.

CHAPTER 3

PRESENTATION AND DISCUSSION OF THE MEASUREMENTS

3.1 Injection through a single hole

3.1.1 Injection through a single hole normal to the mainstream flow*

Figure 16a shows a photograph of the test plate while a summary of the prevailing experimental conditions in the present study is included in table 2, chapter 2.

(1) Flow visualisation

Let us start with the visual observations of the flow for they provide a clear, albeit qualitative, view of the flow structure. Figure 20 shows a plan and side view of the jet for an injectant ratio of 0.24. The jet is made visible by introducing into the secondary jet a mixture of carbon dioxide and vaporised oil. From the side view it is evident that the jet is very rapidly bent over. On the downstream side of the hole, however, there is a region where the jet lifts a little away from the surface, evidenced by the greyish patch behind the hole. The width of the jet increases rapidly at first, then rather more gradually. At a distance of 3 diameters behind the hole the total width of the jet is approximately 2.5 hole diameters. It should be borne in mind that when viewing the jet in plan view, the perceived width is the maximum width of the jet and not necessarily the width at the surface.

Figure 21 provides photographs of the test plate surface after coating it with a solution of saturn yellow pigment with white spirit and kerosene. Figure 21a shows the flow pattern obtained when the jet velocity is 10% of the mainstream. Clearly visible is the development of a wake downstream from the hole. The two lobes adjacent to the downstream side of the injection hole mark a region of separated flow extending approximately half a diameter downstream. A similar pattern is displayed for $M=0.046$ (not shown) though the two lobes then extend

* Part of the present results have been reported by Bergeles et al (45) however, they are included here for completeness.

only 0.3D behind the hole. The corresponding distribution for a dimensionless blowing rate of 0.24 is shown in Fig. 21b. Several features are the same as before: the spread of the wake downstream of the hole and the two separated flow lobes abutting the downstream side of the hole. The region of separated flow has, not unexpectedly, lengthened. To some extent the injectant fluid affects the main flow rather as a blister attached to the surface*. The greater the normalised injection rate the blunter the "equivalent" blister and hence the more extensive the recirculation zone.

A feature of Fig. 21b that could hardly be seen at the lower injection rate is the deflection of the flow upstream of the hole and the associated bending of the vortex lines around the hole. It is entirely expected that this vortex should become more pronounced as the blowing rate is increased since then, as explained above, the injectant stream acts as a progressively blunter obstacle in the path of the external stream. There is just perceptible on Fig. 21b a "pinching" in of the vortex downstream of the hole. The behaviour can be seen more clearly in Fig. 21c. Here the injection rate was the same as Fig. 21b but the white spirit/kerosene mixture was coated only in a patch upstream of the injection hole. The character of the near-surface flow structure indicated by these photographs is quite different from that found by similar surface-visualisation techniques when M is well in excess of unity, Margason and Fearn (46) and McMahon and Mosher (47). In that case, the jet does not reattach, there is strong entrainment of the primary flow into the jet on all sides and the boundaries of the wake region behind the jet are much sharper than in the present study.

* The analogy is not complete however for it allows no recognition of the streamwise vorticity added to the flow by injection.

Finally, figs 2ld and 2le show two photographs of the coated test plate for the case where the solution was initially spread only downstream of the hole. Figure 2le shows the final pattern after 5 minutes; it is virtually identical to that in Fig. 2lb. Figure 2ld shows the pattern at an intermediate time; from this photograph it is evident that the reverse flow region in fact extends further downstream than the limits of the two lobes suggest. A number of different runs were made with the coating beginning at different positions behind the hole; these indicated that for a normalised injection rate of 0.24 the region of reverse flow extended approximately 2 diameters downstream from the hole, with a lateral width of one diameter.

(2) Surface static pressure distribution

Figure 22 shows the distributions of static pressure in the vicinity of the injection hole for values of injection ratio $M=0.1$, 0.24 and 1.0 plotted as contours of the pressure coefficient C_p , which is defined as

$$C_p \equiv (P - P_{ref}) / \frac{1}{2} \rho_j u_\infty^2 v_j$$

where p is the local pressure and P_{ref} is its value in the undisturbed region far upstream of the hole.

The reason for this choice was that it brings the contours for the two lower blowing rates into close agreement - certainly much closer than if the jet velocity or the free-stream velocity alone has been used*. The shapes of the contours have a similar qualitative appearance to those presented by Vogler (9) for blowing rates in the range $1.0-5.0$. Figure 22c shows also comparison between present measurements and those by Vogler (9) for $M=1.0$. It is seen that downstream of the hole the agreement between the two sets of measurements is very good, however upstream of

* In the limit however, as M becomes very large the surface pressure field may be expected to become independent of v_j ; the isobars normalised by $\rho_\infty u_\infty^2$ should then be independent of M .

the hole Vogler's measurements suggest a higher upstream pressure transmission; this difference can be attributed to the difference in boundary layer thickness if Vogler's experiments were for a thicker boundary layer (the boundary layer thickness was not reported).

As regards the present results and taking as the smallest significant value of the pressure coefficient 0.1 it may be concluded that:

- (i) the effects of injection extend approximately $1\frac{1}{2}$ diameters upstream;
- (ii) the maximum value of the pressure coefficient is approximately 1.0 in each case.
- (iii) negative values of pressure coefficient as low as -2.0 occur around the downstream edge of the hole (for the two lower blowing rates).
- (iv) the rate of pressure recovery downstream is substantially more gradual than the rise upstream*.
- (v) lateral effects of the pressure field are more substantial on the downstream side of the hole, the contour of $C_p = -0.1$ extending approximately $2\frac{1}{2}$ diameters off the centre line.

The above behaviour may be explained by regarding the jet as an "obstacle" attached to the surface, which obstructs the upstream flow causing the pressure there to rise; the level of C_p then falls, becoming negative as the deflected fluid accelerates around the hole. Eventually in the downstream region there is a gradual recovery of pressure as the flow reverts back to a normal boundary layer.

Of course the present "obstacle" is a rather special one, whose shape and other characteristics are functions of the injection rate; for example, the emerging jet shears the mainstream flow in a direction opposed to the sense of rotation of a vortex created by the bending of the boundary layer vorticity lines around a blunt object. It is therefore

* The pinching in of the main stream fluid behind the hole noted in fig 21b may be interpreted as a gradual filling-in of the low pressure region behind the hole; or, (equivalently) as a necessary consequence of the diffusion of intense patch of streamwise vorticity associated with the injection itself.

not surprising that the surface pressure distribution turns out to be a function of M . It is however interesting to note that for the two lower blowing rates, the distributions are quite similar when scaled by $\rho_j u_\infty v_j$; this seems to be a general characteristic of discrete jets when they remain substantially attached to the surface, for, as will be seen later, similar behaviour is observed for the case of inclined injection through a hole. It is presumably connected with the fact that $\rho_j u_\infty v_j$ is proportional to the streamwise momentum deficit of the injected fluid.

(3) Velocity measurements

The distribution of mean velocity in the turbulent boundary layer on the plate is shown in Figs 23 and 24. Figure 23 (a-e) show the different kinds of boundary layer measured at five positions along a line parallel with the main flow passing through the hole centre. At $M=0.1$, Fig. 23b there is a large wake-like region behind the hole that gradually fills in with distance downstream. At the higher injection rates shown in Fig. 23 (c-e) the behaviour, is similar except that reverse flow is present (i.e. at $x/D=2.50$ for $M=0.24$). This behaviour bears out the earlier comment that the reverse flow region extends somewhat further downstream than is indicated by the two lobe-shaped patches in Fig. 21b. Measurements for $M=0.1$ and 0.24 have also been taken along a line parallel with the main flow but 0.64 diameters displaced from the hole centre. Figure 24a shows the near-wall profiles for $M=0.24$. The flow close to the wall is progressively decelerated from far upstream to about the mid-hole position; from $0.5 < x/D < 1.4$ however the near-wall flow is accelerated. This behaviour is consistent with the contours of surface static pressure in Fig. 22b along the line $z/D=0.64$ the pressure rises from zero far upstream to a maximum at $x/D=0.4$; there is then a sharp fall in pressure to $x/D=1.4$ and thereafter a gentle rise back to atmospheric pressure. The same qualitative behaviour is exhibited in the profiles of mean velocity for $M=0.1$, fig. 24b

as well for $M=0.046$, Fig 24c though the effects are less pronounced than for $M=0.24$.

The final flow field data refer to the velocity of efflux from the injection hole. Our presumption at the start of the study had been that the mean velocity profile from the injection tube would resemble nearly fully-developed axisymmetric turbulent pipe flow. The very severe pressure gradients around the hole however had suggested that this conjecture might be some way from the truth. Accordingly a hot-wire traverse was made of the flow leaving the injection hole. For $M=0.10$ and 0.24 the profiles, shown in Fig. 25, displayed great non-uniformity. For the lower injection ratio the average velocity over the upstream half of the hole was barely one half that over the downstream half; for the higher injection rate the ratio had fallen to one quarter. It should perhaps be said that the values of velocity plotted in Fig. 25 refer to the total value; this comprises both horizontal and vertical components which we did not attempt to separate. However for $M=0.24$, profile traverses were made at various depths in the hole and these are also shown in Fig. 25. Evidently most of the adjustment in the velocity profile in the tube takes place in the last half diameter prior to discharge. The discovery of the great non-uniformity in the discharge velocity has implications for when one attempts to calculate the flow behaviour.

Turbulence intensity profiles were also measured in the hole for $M=0.1$ and 0.24 , with relative turbulence intensities in excess of 50% recorded over the leading half. Although such high turbulence levels the accuracy of the measurements is quite uncertain. They are nonetheless shown here in Figs 26(a-b). A turbulence intensity profiles with the primary air turned off are also included in the figures; the pipe flow Reynolds number is about 3,500. The profile is typical of those found in developing turbulent pipe flows. It thus appears reasonable to draw direct comparison with measurements by other workers at higher Reynolds

numbers since, provided transition has taken place, the flow structure will be only weakly Reynolds-number dependent.

(4) Wall effectiveness

The distribution of wall film-cooling effectiveness is shown in figure 27 for all the different injection rates studied at four values of z/D . At the smallest injection rate ($M=0.046$) the wall concentrations fall monotonically with increasing x/D and z/D . For $M=0.1$ the general concentration levels are higher and just behind the discharge hole higher values of η are recorded off the centre line than at $z/D=0$. This behaviour becomes more pronounced at the higher blowing rates shown in figures 27c, 27d, 27e. In these cases, even for $z/D=0$, there is an increase in η with x for the first 0.5-1.0 diameter behind the hole. The peak values of effectiveness measured in this immediate neighbourhood of the hole occurred along the line 0.36 diameters displaced from the hole centre. This general behaviour is consistent with the distortion of the jet to a kidney shape by the streamwise vortices induced by injection, see Bergeles et al (48); the centre of the jet is lifted away from the surface but the sides are drawn down in two lobes by the vortex action. Comparison may be made with measurements obtained in the Heat Transfer laboratory of the University of Minnesota - investigations that were mainly concerned with the far-field region. Agreement with the present results is close for $M=0.5$ to $M=1.0$ and also at other blowing rates for positions along $z/D=0$. However, for $M=0.1$ the present data along $z/D=0.64$ lie 20-30% below those of Ramsey and Goldstein (31) and for $M=0.24$ about 10% below (making allowances for the fact that the Minnesota test was for $M=0.20$). One possibly important difference between the experiments was that Ramsey and Goldstein (31) studied heat transfer, not mass transfer. Thus heat conduction in the wall could lead to too high levels of η off the centre line contributing to some extent to the disagreement with the present results.

(5) Conclusions

The main conclusions to emerge from the study of 90° single-hole injection are:

- . Even at injection rates of only 5% of that of the free stream the jet induces a reverse flow region on the downstream side of the hole extending about 0.3 diameters beyond the downstream end of the hole.
- . Surface static pressure measurements show the pressure field to be distorted by the injection approximately $1\frac{1}{2}$ diameters upstream of the hole and $2\frac{1}{2}$ diameters downstream.
- . The static pressure disturbances created by the jet is almost independent of injection rate in the range $M=0.1$ and 0.24 when normalised by $\rho_j v_j u_\infty$, at least for the conditions of the present study.
- . The acceleration and deceleration of the mainstream flow caused by the secondary jet appear to be consistent with the surface static pressure contours.
- . The velocity of ejection is substantially non-uniform due to the rapid variation of static pressure around the hole. For $M=0.24$ only about 25% of the injectant leaves from the leading half of the hole.
- . For injection ratios 0.24 or above the peak effectiveness immediately downstream of discharge occurs off the centreline and increases for a time with x . This pattern is consistent with the jet taking on a kidney shape as it is bent over by the external stream.
- . Where the two studies overlap, very close agreement is obtained with Ramsey and Goldstein data for $M=0.5$ and $M=1.0$. Certain differences emerge, however, in the off-centreline values at lower injection rates, particularly for $M=0.1$.

3.1.2 Injection through a single hole at 30°

Figure 16b shows a photograph of the test plate (all the injection holes except the center one are covered for these tests by thin transparent tape) while a summary of the prevailing experimental conditions is included in table 2, chapter 2.

(1) Flow visualisation

As with 90° injection, we first examine some qualitative information about the flow obtained by spreading upstream of the hole the mixture of yellow pigment and kerosene and then turning on the air supplies. The photograph in Fig. 28a reveals, for $M=0.2$, the surface "streamline" pattern which emerged when the liquid spreads and then evaporates, leaving the yellow pigment. Evidently, the upstream flow is scarcely affected by the injection (presumably because the jet is directed downstream: recall that when the injection angle is 90° , appreciable upstream effects are observed Figs 21), while downstream there exists a clear region where the jet remains substantially attached to the surface. Fig. 28b shows the pattern for $M=0.3$: the behaviour is similar but the lateral extent of the uncovered region has diminished, because although the jet is still attached to the surface, it is ~~straining~~ to lift off. In the light of this behaviour it is not surprising that the flow visualisation photographs for $M=0.5$ and greater (not shown) reveal total coverage of the surface by the pigment, indicating that the jet has lifted off.

Tests with a tracer fluid (black lubrication oil) introduced at varying distances downstream of the hole indicate the presence, for all M , of a small zone of recirculation $0.5D$ wide and $0.3D$ in length (approximate dimensions), which was fed by upstream air entering from the sides. The extent of the recirculation was far smaller than that observed for normal injection at comparable M .

Figures 28c shows some of the results obtained for a blowing rate M of unity. There are evident the reverse flow region (two top left-hand photographs) and the inward-moving fluid towards the downstream edge of the hole (three bottom left-hand photographs) and in the two bottom right hand photographs, the upstream deflection of the mainstream fluid due to the injection.

Finally, the secondary jet was made visible by introducing into it a mixture of carbon dioxide and vaporised oil. Figures 29(a-c) show plan views of the jet for injection ratio $M=0.2, 0.5$ and 1.0 . It should be borne in mind that when observing the jet in plan view, that the perceived width is the maximum width of the jet and not necessarily the width at the surface. The common characteristics of all figures 29(a-c) is the small lateral spread of the jet, the maximum width being around 2.5 hole diameters; the blowing ratio of $M=0.5$ gives the maximum perceived lateral spread of the jet.

Some quantitative information about these and other facets of the flow behaviour on the velocity, pressure and concentration fields will now be given.

(2) Surface static pressure distribution

Figures 30 (a-c) show the distribution of static pressure in the vicinity of the hole for various values of injection ratio M in the range $0.2-1.0$, plotted as contours of the pressure coefficient C_p .

Certain features are common to all plots; thus there is a region of positive pressure starting some distance along the hole and extending laterally and upstream, the distribution suggesting that the peak value of C_p occurs near the leading edge of the hole. Proceeding downstream from this region the pressure rapidly falls to a minimum negative value near the downstream end of the hole and recovers thereafter. The zone of negative C_p also extends laterally and its total extent is generally

greater than the upstream one, if we take the $|c_p| = 0.1$ contour as a measure.

The comparable levels of the pressure coefficients for 30° and 90° injections, mainly for low blowing rates, figures 30 (a-b) and figures 22 (a-b), indicates the success of the choice of the normalising parameter, $\rho_j u_\infty v_j$.

The departure from similarity of the pressure contours downstream of injection hole for blowing rates $M > 0.5$ is attributed to the fact that the jet possesses adequate vertical momentum and penetrates into the mainstream*.

(3) Streamwise velocity distribution

The next series of figures display the development of the streamwise velocity on the center, and in some cases off-center, planes. Typical of the results for the low injection ratios are the data in Fig. 31 for $M=0.2$ (the solid line in this and the remaining figures is the velocity profile which would prevail in the absence of injection, provided for comparison). Referring to the centre-plane development in Fig. 31a, we see that the upstream flow at $x/D = -0.33$ and 0 is mildly retarded by the adverse pressure gradient, while immediately downstream of the hole at $x/D=2.60$ the slow-moving near-wall fluid is that which has just emerged from the injection hole**. Further downstream the action of shear stresses causes the momentum of this fluid to increase (at the expense of that of the surrounding fluid), despite the prevailing adverse pressure gradient. Figs. 31 (b-d) reveal similar, but diminishing effects on planes increasingly remote from the center, due to the smaller amounts of injected fluid which are present there. The disturbance barely extends to the plane at $z/D=0.8$, which is beyond the edge of the hole. As M is increased to 0.3 only slight

* Note that similar behaviour is exhibited the jet for $M=1.0$ at 90° to the mainstream.

** For the present injection angle, the streamwise velocity of the injected fluid is about $0.9Mu_\infty$.

differences are observed, as is shown in Fig. 32. The complete set of measurements are presented later in chapter 5, where they are compared with the predicted values of the developed numerical procedure.

Not surprisingly, the streamwise velocity distributions for $M=0.5$, 1.0 and 1.5 (Figs 33, 34, 35) are different from those for lower M , due to the phenomenon of lift-off (although those for $M=0.5$ share some features with both groups and could therefore be regarded as representing a transitional state), but within themselves they again exhibit strong similarities. Fig. 34 for $M=1.0$ is representative of this group, revealing insignificant upstream effects (because the injection velocity is very nearly that of the boundary layer) and a rather complex downstream structure, initially consisting of three distinct regions identified as the disturbed outer portion of the boundary layer, the detached jet (signalled first at $x/D=2.33$ by the presence of the inner maximum in the velocity profile) and the layer of near-wall fluid beneath the jet. It is interesting to note that although the layer underneath the jet is initially retarded (presumably for the same reasons as at lower M) it subsequently appears to "accelerate" beyond the velocity of the jet, thereby producing another maximum (see $x/D=2.60$ et seq). The reason for this is not obvious (recall that the near-wall pressure gradient is small and in any case adverse) but it may be due to the action of the familiar counter-rotating vortices known to exist in such flows, see Bergeles et al (48), bringing in high velocity fluid from outer layers*. Eventually the layers merge, but the profiles remain far from the undisturbed one even at the further downstream station. At off-center planes the three-layer structure initially persists ($z/D=0.27$) and then reverts to a simpler pattern. The disturbances have essentially disappeared by $z/D=0.8$ (not shown).

* This argument is reinforced by the observation that the peak velocity measured in this region occurs off-center, at $z/D=0.27$.

The centre-plane profiles for $M=0.5$ (fig. 33) and 1.5 (Fig. 35) differ from those above in only minor respects: for example, in the latter case the streamwise velocity of the injected fluid is greater than the free-stream velocity of the boundary layer, hence the occurrence of the maximum velocity within the layer.

(4) Wall film cooling effectiveness measurements

Measured distributions of film-cooling effectiveness are shown for different blowing rates in Figs 36(a-f). The effectiveness is plotted against the downstream distance (x/D) at various lateral locations*. The general trend at the two lower blowing rates $M=0.1$ and 0.2 (Figs. 36a and 36b respectively) is for the effectiveness to decay along lines passing through the hole, and to rise and fall along lines passing nearby: both effects are due to the spreading of the jet and mixing with the mainstream fluid (including that entering the recirculation zone behind the hole). Comparison of the two figures reveals that at this level of blowing, the effect of increasing M is to increase the level and extent of coverage.

Also shown in the above mentioned plots are data drawn from other sources (26, 29) for conditions similar to, but not identical with, the present experiments (cf Table 1). The small discrepancies which are observed may be plausibly attributed to these differences: for example, the tendency for the data of (26) to lie above the present measurements in Fig. 36a is probably due to the lower density ratio and displacement thickness of the former, since in (18)** and (29) it is shown that increases with a reduction in either of these parameters.

* The position of some of these, it should be noted, shifts between $x/D=6.5$ and 8, due to constraints in the construction of the apparatus.

** These data pertain to a row of holes but the spacing:diameter ratio was sufficiently large (3:1) that adjacent jets would not be expected to interact in the region examined here.

The results in Fig. 36c for $M=0.3$ seem at first sight to be similar to those for the lower blowing ratios, but closer inspection reveals the existence of small maxima near the hole for $z/D=0.0, 0.27$. These are presumably a consequence of increased entrainment into the recirculation zone, coupled with the partial lifting off of the jet, which allows more extensive mixing with the mainstream fluid. The maxima may be regarded as defining the points at which the spreading of the jet has allowed it to 'reattach' to the surface, but it must be emphasised that in this instance the 'reattachment line' does not delineate the zone of recirculation. The level and extent of coverage has continued to increase.

As the blowing rate is further raised to $M=0.5$ and beyond, the average level of effectiveness goes through a maximum at $M=0.5$ and then falls, while for the higher blowing rates the local maxima in the profiles become more pronounced and shift downstream. This behaviour is of course the result of the ever-increasing length of the jet trajectory before it reattaches and the consequent dilution of the region beneath by the mainstream fluid. It is also noticeable that, even at $M=0.50$, the off-centre values of η are in some regions equal to or slightly smaller than those for $M=0.3$, indicating that the lateral spread of the jet is no longer increasing: this is caused by the smaller relative velocity between the jet and adjacent fluid, resulting in reduced mixing.

Figs. 36 (d-f) also contain data from other sources, namely (18) (26), (28). The agreement with the present measurements is generally good, the discrepancies being attributable to differences in experimental conditions, notably the upstream boundary layer thickness and the density ratio. In particular, it should be noted that the effect of the former parameter diminishes as M increases, which is in conformity with the findings of the Minnesota group (18, 26, 27, 29, 31).

(5) Practical implications

The practical implications of the present study are perhaps best perceived from Figs. 37a and 37b where the data for all blowing rates have been assembled. Fig. 37a contains the data for the center-line effectiveness which suggest, in agreement with earlier studies, (27, 29) the existence of an optimum value of M of about 0.5 (this assessment only applied, of course to the particular values of pressure gradient, boundary layer thickness etc. employed in those studies). Fig. 37 b shows that this value also yields the maximum lateral coverage, as measured by the paths of the contours of 20% effectiveness. Further, the width of the 20% region is about 1.4 hole diameters, suggesting that the maximum effective spacing between holes should not exceed about $2.8D$ if total coverage is to be achieved.

(6) Conclusions

The main conclusions of the 30° single-hole study may be summarised as follows:

1. The injection process gives rise to appreciable disturbances to the flow field at all injection rates, but, in contrast with 90° injection, the effects are mainly downstream of the hole. Important features of these disturbances are large lateral variations in the streamwise pressure gradient and streamwise recirculation, which are not allowed in conventional analytical and numerical methods of boundary layer analysis.
2. Two modes of behaviour are observed, according to the magnitude of M . For values of about 0.3 and less, the injected jet remains essentially attached to the surface, producing strong surface pressure disturbances which are proportional to the momentum deficit of the injected fluid, as characterised by the quantity $\frac{1}{2} \rho_j v_j u_\infty$, and yielding effectiveness values which rise when M is increased.

3. When M is about 0.5 or above, the jet completely lifts off the surface, allowing penetration of the mainstream fluid beneath, and the length of the trajectory goes up with M . As a consequence the effectiveness now decreases with increasing M , the surface pressure tends to become more uniform and a complex flow structure is produced downstream of the hole.
4. The present results are in good agreement with measurements from other sources and in conformity with earlier findings suggest that for the particular conditions of these measurements the optimum injection rate is about 0.5.

3.2 Injection through a single row of holes at 30°

3.2.1 Presentation and discussion of results

(1) Flow visualisation

We again start with the flow visualisation studies. Figure 38 shows for $M=0.5$ the test section in plan view, with two adjacent jets, made visible by the injection of smoke as in the single hole tests. It is seen that the lateral spread of the jets is very small so that in the near-hole region the shape of the jets is almost identical to that displayed by the single jet in fig.29c. Further downstream however at about $x/D=10$, the jets start interacting with each other. The above findings are in agreement with similar tests performed by Mayle and Camarate (16) (with a larger hole spacing of 8), who found that the integrity of the individual jets is preserved for tens of hole diameters downstream of injection before they interfere with each other.

(2) Static pressure and velocity measurements

Figure 39 shows the distribution of the wall static pressure for blowing rates $M=0.2$ and $M=1.0$.

On the same figure also have been plotted (dotted line) the corresponding measurements for the single hole case, under identical

experimental conditions. Comparison between the two sets of measurements reveals that:

- (i) the level of the pressure coefficients is comparable to single row and single hole cases;
- (ii) injection through a row of holes creates higher pressure upstream of the holes than a single hole case due to the increased blockage of the row of the jets to the mainstream flow;
- (iii) the pressure recovery downstream of the holes is less rapid for the case of a row of holes, the pressure contour for $C_p = -0.1$ extending up to $x/D = 7.0$ downstream in contrast to $x/D = 5.0$ for a single hole.

The measured profiles of streamwise velocity for blowing rates $M = 0.5$ and 1.0 are shown in figures 40 (a-d) along the two planes $z/D = 0.0$ and $z/D = 0.53$. The general behaviour is similar to that for the single hole case which has been discussed at length earlier. The important new features introduced by the presence of adjacent jets is, for $M = 0.5$, as follows:

- At corresponding streamwise locations along the centre line the velocity profiles are closely similar for distances above the surface y/D greater than 0.2 ; however, in the near-wall region ($y/D < 0.2$) the fluid emerging from the single row of holes moves faster than that emerging from a single hole (for example at $x/D = 3.40^*$ and $y/D = 0.07$ the difference in streamwise velocity is about 20%). The difference diminishes with distance downstream.
- Along the $z/D = 0.53$ line there are only slight differences in the velocity profiles for single-row and single-hole angled injection, the fluid from the row of holes always moving slightly faster.

Similar behaviour is found for the higher blowing rate $M = 1.0$ as can

* These observations must be regarded as qualitative, due to the high uncertainties in the velocity measurements in the area immediately downstream of the hole.

be seen in figures 40 (c-d).

The general pattern of the velocity development is in agreement with the measured pressure distribution where it was found that injection from a row of holes creates greater blockage of the upstream flow. This will result in an increased force on the injected fluid which will force it to stay closer to the wall and to move faster.

(3) Film cooling effectiveness measurements

Wall film cooling effectiveness measurements were taken for blowing rates $M=0.25, 0.50, 0.75, 1.0$ and 1.50 and for two different boundary layer displacement thickness $\delta_1/D=0.078$ and $\delta_1/D=0.095$ values most likely to be found in strongly accelerating flows over turbine blades. Liess (20) has also studied a similar range of displacement thickness but he mainly reports averaged film cooling effectiveness measurements, as opposed to the local values which are reported here.

In order to explain the behaviour of the measured wall effectiveness it is helpful to recall the mechanisms, (suggested by the previously reported velocity measurements) by which the injected fluid is diluted by its interaction with the mainstream fluid:

(i) mixing between injected fluid and mainstream fluid passing over the injection hole or entering from the sides downstream of the hole (the rate of mixing depends on the blowing rate).

(ii) entrainment of mainstream fluid into the jet by viscous action.

The rate of mixing here depends on the level of the rates of strain at the mixing interface, which is in turn a function of the relative velocities of jet and main stream fluid.

(iii) transport of low concentration mainstream fluid into the jet by the streamwise vortex pair, whose action is to introduce this fluid into the near-wall region beneath the jet, thereby reducing the effectiveness there.

Let us now examine the film-cooling effectiveness measurements for a displacement thickness $\delta_1/D=0.095$. As for the inclined-injection single-hole measurements of the present study, the data can be divided into two categories, namely those for blowing ratio $M \leq 0.5$ and those for $M > 0.5$, as is evident by the distinctly different behaviour of the centre-line effectiveness downstream of the injection hole for the two cases.

Figure 41a and 41b show the measurements for blowing ratios $M=0.25$ and $M=0.50$. At and near the centre-line, the effectiveness is seen to decrease gradually with downstream distance while at lateral locations for which $z/D > 0.8$ the effectiveness increases very slowly. Immediately downstream of the injection hole the centre-line effectiveness decreases rapidly for blowing rate $M=0.25$ than for $M=0.50$. The lower values for $M=0.25$ are attributed to the smaller amount of coolant introduced into the boundary layer while the increased level of effectiveness for $M=0.50$ at $z/D > 0.8$ is due to the higher rates of strain (i.e. mixing) at these lateral locations.

For blowing ratios greater than M of 0.5 the centre-line effectiveness as well as the lateral spread both decrease with increasing blowing rate. The reason for the decrease of the centre-line effectiveness lies in the penetration of the jet into the mainstream flow while the decrease in the lateral spread is explained by the reduction of the rates of strain between jet and mainstream flow. Flow blowing rates greater than 0.5 the effectiveness immediately downstream of the injection hole and on the centre-line region initially increases with downstream distance, reaches a maximum and then decreases. This phenomenon, which was explained in section 3.1.2, for the single hole case is associated with the lift-off of the jet and the penetration of mainstream fluid beneath it in conjunction with:

- (i) the existence of the local regions of recirculation just downstream

of the hole and

(ii) the actions of the streamwise vortex pairs.

Another consequence of the vortex mechanism is that effectiveness along the $z/D=0.27$ line is higher than $z/D=0.0$. Similar behaviour was noticed for the single-hole angled-injection measurements. For blowing rates $M=1.0$ and 1.50 the overall level of effectiveness decreases in comparison with $M=0.75$ due to the greater penetration of the injected fluid into the mainstream flow. Figure 42 shows the variation of the span-average film-cooling effectiveness with downstream distance for different blowing rates*. It is seen that in the region downstream of the injection hole, a blowing rate of $M=0.5$ is the optimum value, resulting as it does in the highest averaged film cooling effectiveness.

(4) Effect of boundary layer thickness on film cooling effectiveness

The effectiveness measurements for a boundary layer displacement thickness $\delta_1/D=0.095$ presented above were repeated with a smaller displacement thickness $\delta_1/D=0.078$ in order to study the effect of this parameter.

To avoid repetition and since the general behaviour of the measured effectiveness is similar to that for $\delta_1/D=0.095$ the presentation and discussion of the main body of the results will be much abbreviated; the measurements however are shown in figures 43 (a-d).

Let us consider first the qualitative effect of the boundary layer thickness on the effectiveness. In a thinner boundary layer the near-wall fluid possesses higher momentum than the fluid in a thicker boundary layer at the same distance from the wall. Consequently the higher near-wall momentum of the thin boundary layer presses the jet to the surface, thereby increasing the effectiveness.

Figure 44a shows the quantitative effect of δ/D on the effectiveness along the $z/D=0.0$, $z/D=0.8$ and $z/D=1.33$ lines for blowing rate $M=0.25$.

* $\bar{\eta}$ was calculated from η assuming a stepwise lateral variation.

It is seen that the effect is rather small, the smaller boundary layer thickness results in slightly higher effectiveness along $z/D=0.0$ and slightly smaller values along $z/D=1.33$; the reason for this is that for both boundary layer thicknesses the jet remains attached to the surface. When the blowing rate is increased to $M=0.5$ the effect of δ_1/D on η becomes more pronounced; for $\delta_1/D=0.095$ the jet attempts to lift off (this is evident by the small effectiveness maximum behind the hole) whilst for the thinner boundary layer, $\delta_1/D=0.078$, the jet is forced to stay in touch with the surface, the effectiveness maximum behind the hole having disappeared. On the same figure there have been plotted the measurements reported for the single hole, where $\delta_1/D=0.122$. It is seen that the effectiveness along $z/D=0.0$ is lower than for $\delta_1/D=0.095$ while along $z/D=1.33$ is almost the same with than for the single row. The latter feature is coincidental, for it happens that the effect of jet interaction, which would increase η , is offset by the effect of the smaller boundary layer thickness of the single row measurements. Also on the same figure have been plotted measurements of other workers. It is seen that the present measurements for $\delta_1/D=0.078$ are slightly lower than those obtained by Goldstein et al (26) for $\delta_1/D=0.06$, whilst Kadotani's (19) measurements for a thicker boundary layer are lower than the present ones: thus there is general qualitative and quantitative agreement. A conclusion which emerges from this figure is therefore that the displacement thickness exerts an important influence on the effectiveness.

For the higher blowing rate $M=0.75$, figure 44c, it is seen that immediately behind the hole the thinner boundary layer results in substantially increased effectiveness while at lateral locations its effect is rather small. Finally for $M=1.0$, figure 44d, the level of effectiveness along $z/D=0.0$ is higher for the thinner boundary layer while the thicker boundary layer results in higher values along $z/D=1.33$.

Also, the present measurements along $z/D=0.0$ compared with the single hole ones of the present study for a single hole, $\delta_1/D=0.122$, are considerably higher, showing that the jet $\delta_1/D=0.122$ has penetrated substantially further into the mainstream than for $\delta_1/D=0.095$ or $\delta_1/D=0.078$.

In figure 45 the spanwise-averaged effectiveness has been plotted for different blowing rates for $\delta_1/D=0.078$. It can be seen that the blowing rate $M=0.75$ performs better than $M=0.50$ for $x/D > 8.0$ while for $\delta_1/D=0.095$ it was found that $M=0.50$ was the optimum value. It is also appropriate to quote here Kadotani's (19) finding that $M=0.35$ was the optimum blowing rate in his study for displacement thickness of $\delta_1/D=0.175$. We may therefore conclude that the optimum blowing rate depends crucially on the thickness of the boundary layer and it decreases as the boundary layer thickness increases.

Finally figure 46 compares Liess's (20) measurements for averaged effectiveness for $\delta_1/D=0.087$ with corresponding data of the present study. The two sets of measurements are in close agreement.

3.2.2 Conclusions

The main conclusions which may be drawn from the single-row measurements are:

- (i) the upstream pressure effects are slightly higher for a single row than for a single hole, while the pressure recovery downstream of the row of holes is less rapid;
- (ii) the level of effectiveness increases with the blowing rate, reaching a maximum and then it decreases due to the penetration of the jets into the mainstream flow;
- (iii) for the hole spacing examined here $s/D=2.667$, the level of effectiveness along $z/D=1.33$ for a single row and that from a single hole is the same, which provides indirect indication that there is no significant interaction between adjacent jets, at least for $x/D < 10$.

- (iv) decreasing the boundary layer thickness causes the centre-line effectiveness to increase, the off centre-line value to slightly decrease and the laterally-averaged value to increase;
- (v) the boundary-layer thickness significantly affects the wall effectiveness, especially for blowing rates for which the jet is about to lift off from the surface;
- (vi) the optimum blowing rate depends on the boundary layer thickness and increases as the boundary layer thickness decreases (for $\delta_1/D=0.095$, $M=0.5$ while for $\delta_1/D=0.078$, $M_{\text{optimum}} = 0.75$).

Comparison between present measurements and those from other workers reveals good agreement.

3.3 Injection through a double row of holes at 30°

3.3.1 Presenatation and discussion of results

(1) Flow visualisation

We commence by re-examining figure 38 which shows a plan view of the jets emerging from a single row of holes for blowing rate $M=0.5$. Clearly the region between the injection holes extending downstream to about $x/D=10$ would be insufficiently protected in practical turbine-blade cooling situations. Figure 47 shows a plan view of the jets emerging from the double row of holes for blowing rate $M=0.5$ in which three of the jets were made visible by introducing smoke into their supply lines as in the previous studies. Evidently the jet emerging from the first row of holes retains its identity up to $x/D=4.50$ (the half-distance to the second row of holes) where there it contracts slightly and then, at $x/D=5.50$, merges with the jets issuing from the second row, thus resulting in complete coverage of the test plate at locations downstream therefrom. Thus, the jets from the first row of holes cover the region which otherwise would remain unprotected if only a single row of holes had been employed.

(2) Velocity measurements

Figures 48 (a-d) show velocity measurements taken along lines $z/D=1.33$ (centre-line of the second row of holes), 1.07, 0.80 and 0.53. The general behaviour is almost the same with that from a single row of holes: however, there are some differences which are described below.

Along the centre-line of the downstream row of holes ($z/D=1.33$) the velocities are 5% higher than those from a single row of holes especially near the wall. The reason for this behaviour lies in the acceleration of the mainstream fluid along this line due to the blockage effect of the first row of holes (observe the favourable pressure gradient along $z/D=1.33$ in figure 39) and consequently the boundary layer approaching the second row of hole has a higher momentum resulting in greater bending of the injected fluid. Along the other lines, $z/D=1.07$, 0.80 the differences become smaller and in fact along the $z/D=0.80$ line the near-wall fluid moves 5% slower than if only the downstream row of holes existed. This is because, now, the boundary layer approaching the second row of holes has already been decelerated by the injection from the first row of holes. Figure 48d shows more clearly the development of the velocity profiles along the $z/D=0.53$ line and the interaction of the two rows of holes. It is seen that up to $x/D=3.20$ the injection from the second row of holes does not affect the velocity profiles, since they are identical to those if only the first row of holes existed (dotted lines). Further downstream ($x/D > 3.2$) the injection from the downstream row of holes affects the velocity profiles and the near-wall fluid starts to accelerate. It is seen that at $x/D=4.50$ the velocity of the near wall fluid is nearly that which would prevail if only the second row of holes existed (solid line on figure 48d), showing plainly that the second row exerts the predominant influence in this region.

(3) Effectiveness measurements

Effectiveness measurements were taken for blowing rates $M=0.25$, 0.5 , 0.75 and 1.0 for two different boundary layer thicknesses, $\delta_1/D=0.078$ and $\delta_1/D=0.095^*$. These conditions are identical to those for the single row of holes experiments allowing comparisons to be made.

Figure 49 shows the film cooling effectiveness for a blowing rate $M=0.50$ and boundary layer thickness $\delta_1/D=0.095$ (the dotted lines on this and subsequent figures represent the effectiveness under identical experimental conditions for a single row of holes). Let us focus our attention first on the measurements along lines $z/D=0.0$ and $z/D=0.53$. Along the line $z/D=0.0$ the effectiveness decreases at a much higher rate than if the second row of holes did not exist. This characteristic is more pronounced as we move laterally i.e. along the $z/D=0.53$ line the effectiveness drops at a greater rate, reaches a minimum value at about $x/D=4.50$ and recovers fairly quickly obtaining a value which is slightly higher than along $z/D=0.27$ line. The explanation for this behaviour lies in the effect of the second row of holes on the development of jets emerging from the first row; in particular the sideways deflection of the mainstream fluid approaching the second row of holes is responsible for the existence of the effectiveness minimum along $z/D=0.53$ line and the consequent mixing of the second row jets with those from the first row explains the quick recovery of the effectiveness for $x/D > 4.50$ along $z/D=0.53$ line. This mechanism is consistent with the observation that the maximum effect occurs near $z/D=0.53$ line and is smaller along $z/D=0.0$; it also explains the narrowing in the flow visualisation photographs of the jet issuing from the first row of holes in the vicinity of $x/D=4.50$. Also of note in figure 49 is the dotted line which indicates the effectiveness variation along $z/D=1.33$ which would occur if only the second row of holes existed. It is evident that the two sets of measurements coincide at least

*These values of δ_1/D prevail at the upstream edge of the second row of holes. The corresponding values at the upstream edge of the first row are 9% smaller.

in the immediate vicinity of the downstream hole, which is an indication that the upstream row does not significantly influence the jets emerging from the downstream row, at least for $M=0.50$ and $\delta_1/D=0.095$. The mixing between the jets from the two rows proceeds with the downstream distance, and almost uniform effectiveness is achieved at about 20 hole diameters downstream.

The discovery of a region of minimum effectiveness between the injection holes is an unwelcome one because of its practical implications; for on a turbine blade this region would be a hot spot which would create considerable thermal stresses and thereby affecting the life of the blade.

Figure 50 shows the effectiveness measurements for a blowing rate $M=0.25$. The general characteristics to those for $M=0.5$. The main differences are that the effects of the second row on the first row are smaller since the lower blowing rate gives rise to smaller upstream disturbances. Also the jets themselves have smaller lateral spread; consequently the minimum value of effectiveness along $z/D=0.53$ is higher than for $M=0.50$, the effect on centre-line effectiveness $z/D=0.0$ is smaller, and the rate of recovery along $z/D=0.53$ is less.

For higher blowing rate, $M=1.0$ (figure 51), the effect of the second row on the effectiveness along $z/D=0.53$ is as expected higher since the upstream effects, are stronger. However, there is no appreciable effect on the effectiveness along $z/D=0.0$ (in contrast with the behaviour for $M=0.5$), while, the first row of holes does now exert significant influence on the second row. The reason for this new effect lies in the different behaviour of the individual jets for this blowing rate. Thus, the single-row experiments showed that the jets emerging from the holes lift off the surface and penetrate into the main stream, the level of effectiveness near the center line ($z/D=0.0$) being controlled by diffusion from jet fluid above the surface (jet "reattachment"). In

the double row case the jets from the first row presumably lift off the surface and force the jets emerging from the second, to remain nearer to the surface thus resulting in higher effectiveness. This effect is expected to be more pronounced at the lateral location $z/D=0.80$ and less pronounced on $z/D=1.33$ as is confirmed in figure 51. Due to the intense mixing, lateral uniformity is almost achieved 20 hole diameters downstream of the second row.

Figure 52 shows the variation of the laterally-averaged effectiveness with downstream distance for the various blowing rates (the averaged effectiveness was calculated from the present measurements in the manner described earlier). Evidently in the immediate vicinity of the downstream row of holes $M=0.5$ yields higher effectiveness, than $M=0.25$ or $M=1.0$; however further downstream ($x/D > 9.0$) $M=1.0$ provides maximum effectiveness. This result, which is a consequence of reduced penetration into the mainstream of the second row of jets, may be contrasted with the conclusion drawn from the single row injection study that a blowing rate of $M=0.50$ performed better than $M=1.0$ under identical experimental conditions. On the same figure 52 are also plotted for comparison results obtained from the single row study. Evidently the staggered double row geometry results in better film cooling for a given mass flow of coolant than the single row geometry: this can be seen from comparison of the averaged effectiveness for $M=0.50$ for the single row with that for $M=0.25$ for the double row (the amount of coolant introduced into the boundary layer being the same in the two cases). The same conclusion is drawn if we compare the curve for $M=1.0$ for the single row with that for the double row for $M=0.50$.

(4) Effect of boundary layer thickness

We will now examine the data for a displacement thickness $\delta_1/D=0.078$ which is 18% smaller than that of the experiments described above.

Figure 53 shows the measurements for a blowing rate $M=0.50$. The following are the main differences between these data and those for $\delta_1/D=0.095$ (figure 49). There is hardly any effect along $z/D=0.0$ of the second row of holes (the dotted line represents corresponding measurements for single row); likewise, along $z/D=1.33$ there is scarcely any influence of the first row. The minimum effectiveness along $z/D=0.53$ is higher for the thinner boundary layer due to the smaller lateral deflection of the mainstream fluid by the second row. The behaviour of the effectiveness for the lower blowing rate $M=0.25$ is similar to the above but smaller differences along the $z/D=0.53$ occur for the two boundary layer thickness, figure 54. For a blowing rate $M=1.0$, figure 55, the jets from the second row are forced to stay closer to the surface resulting in higher levels of effectiveness (the solid line represents the corresponding measurements for the thicker boundary layer $\delta_1/D=0.095$).

The laterally-averaged film cooling effectiveness is shown on figure 56 for the different blowing rates for both displacement thickness. It is clear that the blowing rate $M=1.0$ performs better than $M=0.25$ or $M=0.50$. Comparing the averaged effectiveness for the two different displacement thickness it is concluded that for blowing rates $M=0.25$ and $M=1.0$ the effectiveness are almost the same for the two thickness while for $M=0.5$ the thinner boundary layer gives on the average 10% higher mean effectiveness.

3.32 Conclusions

To summarise, the main conclusions of the double row study:

- (i) the presence of the second row of holes causes the appearance of a region of minimum effectiveness located between the centrelines of the two rows of holes, just downstream of the leading edge of the second row of holes ($x/D=4.50$ and $z/D=0.53$).

- (ii) for blowing rate $M=1.0$ the jets emerging from the second row are forced by the jets from the first row to stay nearer to the surface, thus resulting in higher effectiveness.
- (iii) the blowing rate $M=1.0$ yields higher overall effectiveness than $M=0.25$ and $M=0.50$.
- (iv) the double row cooling geometry is superior to a single row for a given mass flow of coolant.
- (v) decreasing the boundary layer thickness only slightly affects the mean effectiveness for $M=0.25$ and $M=1.0$ whilst for $M=0.5$ is increased by an average of 10%.
- (vi) almost uniform wall effectiveness prevails for all blowing rates by about 20 hole diameters downstream of injection.

3.4 Conclusions of the present experimental study

Below are the main overall conclusions to be drawn from the present experimental study (strictly valid, it should be emphasised, only for the conditions of these experiments).

3.4.1 Hydrodynamic field

- (1) In all cases the pressure field created by injection spreads approximately $1\frac{1}{2}$ hole diameters upstream of the hole and extends a greater distance (3-5 diameters) downstream (the actual distance depending on the geometry).
- (2) For a given angle of injection, the pressure field becomes almost similar for various blowing rates (especially for those for which the jet remains attached to the surface) if non-dimensionalised with the product of injection and free-stream velocities.
- (3) The secondary velocity profile at the exit of the hole is highly distorted by the mainstream flow, with in some circumstances more than

half of the secondary mass flow emerging from the downstream quarter of the hole (this conclusion holds for $M=0.24$, $\alpha=90^\circ$, the actual jet velocity profile depends on the blowing rate M);

- (4) Behind the hole there is a region of reverse flow whose dimensions depend on the angle of injection and the blowing rate. For inclined injection at $\alpha=30^\circ$, this region is very small indeed, being approximately $0.3D$ in length, $0.5D$ in width and $0.05D$ in height and attached to the downstream edge of the hole).
- (5) There is a blowing rate above which the jet lifts off the surface and mainstream fluid penetrates underneath it; the lift-off value depends on the thickness of the boundary layer and for $\alpha=30^\circ$ is: $M=0.3$ for $\delta_1/D=0.122$, $M=0.5$ for $\delta_1/D=0.095$ and $M=0.75$ for $\delta_1/D=0.078$.
- (6) The existence of the vortex pair behind the hole results in mainstream fluid being transported beneath the jet with results that the near wall fluid may move faster than the jet itself.

3.4.2 Concentration field

- (1) For a hole spacing of $S/D=2.667$ and $\alpha=30^\circ$ there is no significant interaction between adjacent jets up to $x/D=10$.
- (2) The existence of the counter-rotating vortex pair behind the hole causes the maximum effectiveness to occur at an off-centre-line location (i.e. at $z/D=0.27$ for 30°).
- (3) The effectiveness increases with the blowing rate up to a maximum (M_{optimum}) and then decreases for higher blowing rates, this behaviour being associated with the lift off the jet.
- (4) Normal injection results in higher level of effectiveness at lateral locations beyond $z/D=0.6$ but gives smaller levels in the region of centre-line.

- (5) The boundary layer thickness is a significant parameter in film cooling: decreasing it results in increased centre-line and laterally-averaged effectiveness.
- (6) The optimum blowing rate depends on the boundary layer thickness and increases as the boundary layer thickness decreases.
- (7) A double-row cooling geometry results in better film cooling coverage than a single row for the same amount of mass coolant injected into the mainstream flow.
- (8) The optimum blowing rate for a double row is higher than the corresponding one for a single row, because of the effect of the first row of holes in forcing the jets from the second row, to remain closer to the surface.
- (9) The presence of the second row of holes causes the appearance of a region of minimum effectiveness located between the centre lines of the two rows of holes just downstream of the leading edge of the second row.
- (10) Almost uniform wall effectiveness prevails for all blowing rates by about 20 hole diameters downstream of injection.
- (11) The present results with measurements from the other sources are in good quantitative agreement.

CHAPTER 4

THEORETICAL STUDY

4.1 The Mathematical foundation of the problem

4.1.1 Introduction

This section presents a mathematical statement of the physical laws which govern the flow field created by the interaction of discrete jets with a mainstream flow. The form of the equations for general three dimensional flows is presented first and then the special simplified equations are derived which are held to characterise the class of discrete jets under study here. The section also sets out the inlet and boundary conditions of the flow field.

4.1.2 The Equations

The physical laws governing the flow field are the conservation principles of mass, momentum and energy; the mathematical form of these laws expressed in Cartesian tensor form notation, is, (Batchelor (49)):

$$\frac{D\rho}{Dt} + \rho \frac{\partial u_j}{\partial x_j} = 0 \quad (\text{continuity}) \quad (4.1)$$

$$\rho \frac{Du_i}{Dt} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} (\mu e_{ij}) \quad (\text{momentum, } i=1,2,3) \quad (4.2)$$

$$\rho \frac{Dh}{Dt} = \frac{\partial}{\partial x_j} (k \frac{\partial T}{\partial x_j}) + \frac{\partial}{\partial x_j} (u_k \mu e_{kj}) \quad (\text{energy}) \quad (4.3)$$

where:

- (i) u_i is the instantaneous velocity
- (ii) e_{ij} is the rate of strain tensor equal to

$$e_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \quad (4.4)$$

- (iii) h is the total specific energy defined as:

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

- (iv) C_p is the specific heat of fluid under constant pressure
- (v) δ_{ij} is the kronecker delta
- (vi) $\frac{D}{Dt}$ is the total derivative defined as

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + u_j \frac{\partial}{\partial x_j} = \frac{\partial}{\partial t} + u \frac{\partial}{\partial x} + v \frac{\partial}{\partial y} + w \frac{\partial}{\partial z} \quad (4.6)$$

The density of the fluid is obtained from the equation of state

$$\rho = \rho(P, T) \quad (4.7)$$

Equations (4.1) - (4.3) and (4.7) form a closed set of six elliptic differential equations with six unknowns, the three velocities (u,v,w) the pressure, the temperature and the density of the fluid. The co-ordinate system is shown in figure 57a.

Despite the simplicity of the physical laws, their mathematical expressions are very complicated and hence the equations are very difficult to solve. This is due to the non-linear character of the of the convection part of the equations (4.2) (the left hand side of that equation expressing the acceleration of the fluid element) and also to the strong coupling among the equations.

4.1.3 The equations for a quasi-orthogonal co-ordinate system

In the discrete-hole cooling (DHC) geometry the mainstream fluid is deflected upwards and sideways as it approaches the injection holes; consequently the streamlines are bent and effect of injection spreads gradually outwards into the mainstream flow. Thus it seems that a quasi-orthogonal co-ordinate system that would expand so as to conform approximately with the development of the boundary layer profile would be suitable for the study of the present problem (such a coordinate system minimises the number of the required grid nodes and the false diffusion associated with the numerical solution of the equations).

In the (ξ, n, ζ) curvilinear system adopted, figure 57b, the n, ζ directions are mutually orthogonal, the (n, ζ) plane being defined as normal

to the surface and form a rectilinear system, while the third co-ordinate ξ is allowed to depart from orthogonality with the (n, ζ) planes to within specified limits, so as to align more closely with the flow field.

The co-ordinates ξ, n, ζ are related to the Cartesian coordinates (x, y, z) by

$$\begin{aligned}\frac{\partial}{\partial x} &= \frac{\partial}{\partial \xi} \frac{\partial \xi}{\partial x} + \frac{\partial}{\partial \eta} \frac{\partial \eta}{\partial x} + \frac{\partial}{\partial \zeta} \frac{\partial \zeta}{\partial x} \\ \frac{\partial}{\partial y} &= \frac{\partial}{\partial \xi} \frac{\partial \xi}{\partial y} + \frac{\partial}{\partial \eta} \frac{\partial \eta}{\partial y} + \frac{\partial}{\partial \zeta} \frac{\partial \zeta}{\partial y} \\ \frac{\partial}{\partial z} &= \frac{\partial}{\partial \xi} \frac{\partial \xi}{\partial z} + \frac{\partial}{\partial \eta} \frac{\partial \eta}{\partial z} + \frac{\partial}{\partial \zeta} \frac{\partial \zeta}{\partial z}\end{aligned}\tag{4.8}$$

wherein

$$\xi \equiv x / \cos \theta, \quad \eta \equiv y / \delta(x), \quad \zeta \equiv z\tag{4.9}$$

and $\delta(x)$ is the local height of the domain of integration, taken as constant in the spanwise direction.

From relations (4.9) we obtain,

$$\begin{aligned}\frac{\partial \xi}{\partial x} &= \frac{1}{\cos \theta} - \frac{x}{\cos^2 \theta} \sin \theta \frac{\partial \theta}{\partial x}, \quad \frac{\partial \eta}{\partial x} = -\frac{y}{\delta^2} \frac{d\delta}{dx} \\ \frac{\partial \xi}{\partial y} &= \frac{1}{\cos^2 \theta} \sin \theta \frac{\partial \theta}{\partial y}, \quad \frac{\partial \eta}{\partial y} = \frac{1}{\delta}, \quad \frac{\partial \zeta}{\partial z} = 1\end{aligned}\tag{4.10}$$

If we assume that:

$$\frac{\partial \theta}{\partial x} = O(\theta) \text{ (order of magnitude } \theta)$$

$$\frac{\partial \theta}{\partial y} = O(\theta) \cdot O\left(\frac{1}{\delta}\right)$$

$$\frac{\partial \delta}{\partial x} = O(\theta)$$

equations (4.10) become after ignoring terms of $O(\theta^2)$:

$$\frac{\partial \xi}{\partial x} \approx 1, \quad \frac{\partial \eta}{\partial x} = -\frac{y}{\delta^2} \frac{d\delta}{dx} \quad (4.11)$$

$$\frac{\partial \xi}{\partial y} = 0, \quad \frac{\partial \eta}{\partial y} = \frac{1}{\delta}, \quad \frac{\partial \zeta}{\partial z} = 1$$

where the substitution of $\cos \theta \approx 1$ sets an upper limit to the rate of grid expansion of

$$\theta_{\max} = 18^\circ \quad (4.12)$$

dictated by the requirement that $\cos \theta_{\max} \approx 1$ to within 5% accuracy.

Consequently relations (4.8) may now be written as

$$\begin{aligned} \frac{\partial}{\partial x} &= \frac{\partial}{\partial \xi} - \eta \frac{1}{\delta} \frac{d\delta}{dx} \frac{\partial}{\partial \eta} \\ \frac{\partial}{\partial y} &= \frac{1}{\delta} \frac{\partial}{\partial \eta} \end{aligned} \quad (4.13)$$

$$\frac{\partial}{\partial z} = \frac{\partial}{\partial \zeta}$$

It is evident from the relations (4.13) that in transforming (4.1), (4.2) and (4.3) into the (ξ, η, ζ) coordinate system extra terms would appear. For brevity, the presentation of the form of equations (4.1), (4.2) and (4.3) for the (ξ, η, ζ) coordinate system will be deferred until certain simplifications are made which will be explained below.

4.1.4 Some simplifications for the DHC geometry

(1) The time dependency of the equations

The time dependent character of the flow is due to its turbulent nature; that is to the existence of eddies which rotate at various frequencies and are convected and diffused in the flow field. The time scale involved in these velocity changes is much smaller than the time scales which the engineer is concerned in practical situation. The time mean values of the velocity and thermal fields usually suffice for practical engineering design; consequently equations (4.1), (4.2) and (4.3) are usually time integrated and reduced to a less complex steady state form.

This practice will be followed here.

(2) Importance of compressibility effects

In many practical situations the rate at which the density of a material element changes is often negligibly small; in these circumstances the mass conservation equation (4.1) reduces to

$$\frac{\partial \rho}{\partial x_j} = 0 \quad (4.14)$$

and the rate of strain tensor to the simplified form

$$e_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \quad (4.15)$$

For flow over a turbine blade, however, the density of the fluid changes appreciably because of pressure and temperature gradients along the surface of the blade and across the blade boundary layer, the latter due namely to the temperature differences between the injected coolant and mainstream fluid. The above simplifications are not therefore appropriate in the present circumstances; however, the maximum Mach number examined in the present study was about 0.4; so it is justified to neglect pressure effects on density.

The contribution of the shear work in the equation for conservation of energy (4.3), was neglected because the maximum Eckert number occurred in the present study was of the order of 0.15; likewise the kinetic energy of the fluid was omitted from the enthalpy equation.

It should be emphasised that the solution procedure to be described is capable of handling all of the above phenomena, should it prove to be necessary.

(3) Boundary layer assumptions

As was mentioned in chapter 1 the aim of DHC is for the jets emerging from the injection holes to remain in touch with the surface to be protected. To achieve this aim requires that either that the velocity of the jets be kept low or that they must emerge from the holes obliquely to

to the mainstream. In either case, it initially appeared reasonable to assume that there was no region of reverse flow within the flow field under investigation*.

This basic assumption of the existence of a predominant flow direction (PFD) in conjunction with the assumption that the Reynolds number based on u_∞ and δ is large (which is valid in practical situations) allows a boundary-layer type of analysis.

The usual assumptions associated with a boundary-layer flow are, (50):

- (i) There is a predominant flow direction in the flow field (taken here to be the x_1 direction)
- (ii) The rate of change of diffusion fluxes of momentum and energy along the PFD are negligible and
- (iii) The downstream pressure field has negligible influence on the upstream flow conditions.

Mathematically, assumptions (i) and (ii) are expressed as

$$u_1 > 0; \quad \frac{\partial}{\partial x_k} \left(\Gamma_\phi \frac{\partial \phi}{\partial x_\ell} \right) = 0 \quad (4.16)$$

where ϕ stands for u_i or T , either of the indices k and ℓ equals 1 and Γ_ϕ is the diffusion coefficient for the ϕ variable.

To examine the conditions under which assumption (iii) is valid let us study the Poisson equation for pressure in an incompressible flow which is obtained by taking the divergence of the u_i momentum, equation (4.2), and rearranging. The result is:

$$\frac{\partial^2 p}{\partial x_i^2} = -\rho \frac{\partial}{\partial x_i} \left(u_j \frac{\partial u_i}{\partial x_j} \right) \quad (4.17)$$

* Within the course of the investigation, however, it was determined that this assumption was not entirely correct and weak reverse flow did occur. Consequently a special treatment was developed to handle the weak reverse flow region within the framework of a boundary layer equations.

Assumption (iii) above is valid if the second derivative in the PFD are negligible in equation (4.17), in which case, equation (4.17) becomes parabolic in the x_1 direction*. From the right hand side of equation (4.17) we deduce that assumption (iii) is correct if**

$$\frac{\partial^2 u_1}{\partial x_1^2} < < \frac{\partial^2 u_2}{\partial x_2^2} \quad \text{or} \quad \frac{\partial^2 u_2}{\partial x_1 \partial x_2} \quad \text{etc} \quad (4.18)$$

and on dimensional grounds

$$\frac{u_1}{D^2} < < \frac{u_2}{\delta^2} \quad (4.19)$$

where D is a typical length in the x_1 direction.

For thin shear flows and in the absence of injection (e.g. flow over flat plate) condition (4.19) applies, because $u_2 \approx O(D^{-0.8})$ and $\delta = O(D^{0.2})$ Ref (50). Consequently equation (4.18) is valid and downstream flow effects do not transmit upstream. However for the case of injection through discrete holes and in the vicinity of the injection hole,

$$u_2 \approx O(u_1) \quad \text{and} \quad \delta \approx O(D)$$

Consequently relation (4.18) is not valid and the assumption that upstream conditions are not influenced by downstream pressure events is not justified. However at some distance downstream of the location of injection it may be expected that the pressure equation will take on a parabolic character.

4.1.5 The simplified form of the equations on the (ξ, n, ζ) coordinate system

By introducing the earlier assumptions into the set of equations (4.1), (4.2) and (4.3) expressed in the (ξ, n, ζ) coordinate system we obtain the following equations (where u, v, w, p, T are now time mean values)

* Also if $\partial p / \partial x_1$ is constant at each cross-stream plane and known.

** Relation (4.18) is one possible sufficient condition for equation (4.17) to be parabolic.

$$\frac{\partial}{\partial \xi} (\rho u) + \frac{1}{\delta} \left(\frac{\partial}{\partial \eta} (\rho v) - \eta \frac{d\delta}{dx} \frac{\partial (\rho u)}{\partial \eta} \right) + \frac{\partial}{\partial \xi} (\rho w) = 0 \quad (4.20)$$

$$\begin{aligned} \frac{\partial}{\partial \xi} (\rho uu) + \frac{1}{\delta} \left(\frac{\partial}{\partial \eta} (\rho vu) - \eta \frac{d\delta}{dx} \frac{\partial (\rho uu)}{\partial \eta} \right) + \frac{\partial}{\partial \xi} (\rho wu) = \\ - \frac{\partial p}{\partial \xi} + \frac{1}{\delta} \frac{\partial}{\partial \eta} \left(\mu_{n,eff} \left(\frac{1}{\delta} \frac{\partial u}{\partial \eta} \right) \right) + \frac{\partial}{\partial \xi} \left(\mu_{\zeta,eff} \left(\frac{\partial u}{\partial \xi} \right) \right) \\ - \eta \frac{1}{\delta} \frac{d\delta}{dx} \frac{\partial p}{\partial \eta} \end{aligned}$$

$$\begin{aligned} \frac{\partial}{\partial \xi} (\rho uv) + \frac{1}{\delta} \left(\frac{\partial}{\partial \eta} (\rho vv) - \eta \frac{d\delta}{dx} \frac{\partial (\rho uv)}{\partial \eta} \right) + \frac{\partial}{\partial \xi} (\rho wv) = \quad (4.21) \\ - \frac{1}{\delta} \frac{\partial p}{\partial \eta} + \frac{1}{\delta} \frac{\partial}{\partial \eta} \left(\mu_{n,eff} \left(2 \frac{1}{\delta} \frac{\partial v}{\partial \eta} \right) \right) + \frac{\partial}{\partial \xi} \left(\mu_{\zeta,eff} \left(\frac{\partial v}{\partial \xi} + \frac{1}{\delta} \frac{\partial w}{\partial \eta} \right) \right) \end{aligned}$$

$$\begin{aligned} \frac{\partial}{\partial \xi} (\rho uw) + \frac{1}{\delta} \left(\frac{\partial}{\partial \eta} (\rho vw) - \eta \frac{d\delta}{dx} \frac{\partial (\rho uw)}{\partial \eta} \right) + \frac{\partial}{\partial \xi} (\rho ww) = \\ - \frac{\partial p}{\partial \xi} + \frac{1}{\delta} \frac{\partial}{\partial \eta} \left(\mu_{n,eff} \left(\frac{1}{\delta} \frac{\partial w}{\partial \eta} + \frac{\partial v}{\partial \xi} \right) \right) + \frac{\partial}{\partial \xi} \left(\mu_{\zeta,eff} \left(2 \frac{\partial w}{\partial \xi} \right) \right) \end{aligned}$$

$$\begin{aligned} \frac{\partial}{\partial \xi} (\rho uC) + \frac{1}{\delta} \left(\frac{\partial}{\partial \eta} (\rho vC) - \eta \frac{d\delta}{dx} \frac{\partial (\rho uC)}{\partial \eta} \right) + \frac{\partial}{\partial \xi} (\rho wC) = \quad (4.22) \\ \frac{1}{\delta} \frac{\partial}{\partial \eta} \left(\Gamma_{n,eff} \frac{1}{\delta} \frac{\partial C}{\partial \eta} \right) + \frac{\partial}{\partial \xi} \left(\Gamma_{\zeta,eff} \frac{\partial C}{\partial \xi} \right) \end{aligned}$$

where $\mu_{n,eff}$, $\mu_{\zeta,eff}$ and $\Gamma_{n,eff}$, $\Gamma_{\zeta,eff}$ are effective diffusion exchange coefficients for momentum and heat defined later in the section on turbulence modelling and C is a non-dimensional variable for temperature defined as

$$C = (c_p T - c_{p\infty} T_{\infty}) / (c_{p_j} T_j - c_{p\infty} T_{\infty}) \quad (4.23)$$

Equations (4.20), (4.21), (4.22) with the equation of state and the equations calculating the exchange coefficients Γ_{eff} and Γ_{eff} are solved in the present study with the following initial and boundary conditions.

4.1.6 Initial and boundary conditions

For the full specification of the Mathematical problem, boundary as well as initial conditions are required. They are summarised in table 3, below for the case of jets emerging from a row of discrete holes. The domain of integration is shown in figure 57c; it is supposed that the downstream boundary is a region where the flow has reverted back to a parabolic character, or nearly so.

boundary	u	v	w	P	c
bottom(plate) face	0	0	0	-	$\frac{\partial c}{\partial y} = 0$
at hole exit	$u_e = \text{const}$ $u_e = \text{linear}$ $P_{tot} = \text{const}$	$v_e = \text{const}$ $v_e = \text{linear}$	0	-	1
top(free stream) face	$u_\infty = u(x)$	-	0	$P_\infty(x)$	0
upstream face	$u = u(y)$	$v = v(y)$	0	-	0
downstream face	-	-	-	$\frac{\partial P}{\partial x^2} = 0$ $\frac{\partial P}{\partial x} = 0$	-
edges	$\frac{\partial u}{\partial z} = 0$	$\frac{\partial v}{\partial z} = 0$	0	$\frac{\partial P}{\partial z} = 0$	$\frac{\partial c}{\partial z} = 0$

Table 3: Summary of initial and boundary conditions

As seen in the table various boundary conditions were applied over the injection hole. The reason is that in practical situations the secondary flow is supplied through a pipe and the interaction with the mainstream flow is bound to create a non-uniform distribution of static pressure over the injection hole which will affect the velocity distribution in the fluid emerging from the hole. Consequently, the velocity profile of the secondary flow will not in fact be uniform as boundary condition (i) of table 3 supposes*. Indeed the profile will probably be distorted to a degree which will depend on the blowing rate, as measurements of inlet plane velocity profiles reported in chapter 3 suggest; consequently boundary conditions (ii) and (iii) were also applied as alternatives to (i) over the injection hole.

At the top face (free stream) the streamwise velocity is prescribed as well the pressure while at the upstream face a 2-dimensional turbulent boundary layer velocity profile is assumed hence the streamwise and vertical velocities are prescribed.

Since the pressure field in the vicinity of the injection hole is elliptic, there is need to apply boundary conditions for pressure at the downstream boundary of the calculation domain. Two conditions were tried at this location

$$(i) \quad \frac{\partial^2 p}{\partial x^2} = 0$$

$$(ii) \quad \frac{\partial p}{\partial x} = 0$$

while both conditions led to almost the same results for low blowing rates, boundary condition (ii) was found to be superior for high blowing rates.

* A fully turbulent pipe flow velocity profile would to the same extent deviate from the real velocity profile.

In the region well downstream from the hole the pressure equation may be regarded as parabolic and no downstream boundary condition on the pressure is required.

Finally the wall is assumed to be adiabatic, consequently the heat flux throughout is zero, $\frac{\partial C}{\partial y} = 0$.

4.2 Mathematical modelling of Turbulence

4.2.1 Introductory remarks

We start the presentation of the work on mathematical modelling of Turbulence with a definition quoted from the text book by Bradshaw (51).

"Turbulence is a three dimensional time dependent motion

in which vortex stretching causes velocity fluctuations to spread to all wave lengths between a minimum determined by viscous forces and a maximum determined by the boundary conditions of the flow".

In view of this feature, it is evident that turbulent flow is represented by the time-dependent 3-dimensional form of the Navier-Stokes continuity and energy equations, (4.1), (4.2) and (4.3).

In principle, it is possible to devise a numerical algorithm to solve equations (4.1) - (4.3) by digital computer. However a difficulty arises because of the limited storage capabilities of the present day computers (or even those of the next generation). The unsteady character of the turbulent flow is mainly due to the existence of rotating eddies most of which have dimensions an order of magnitude or more smaller than any typical length of the geometry of the flow field. For an adequate numerical representation of these eddies the number of grid points for which equations (4.1) - (4.3) would require to be discretized and solved is very large, indeed well beyond the storage capacity of existing computers.

This difficulty has led to the application of statistical representations of turbulent flows. The main concept in this approach is that the time dependent flow can be represented by a mean flow upon which is superimposed a time dependent velocity field whose time mean velocity at each point of the flow field is by definition zero.

From the engineering point of view the quantities of the flow field which are of interest are the mean quantities (i.e. mean velocity, mean pressure, mean heat transfer rate etc). Consequently in this section we are concerned with methods of substituting time-dependent Navier Stokes equations by ones that are suitable for calculating the steady state flow field.

4.2.2 The steady-state momentum energy and continuity equations for turbulent flow

Formalising the above remarks, the instantaneous velocity, and pressure are expressed as:

$$U_i = u_i + u'_i, \quad P = p + p'$$

where

$$u_i = \int_{t=0}^{\infty} u_i \, dt, \quad p = \int_{t=0}^{\infty} P \, dt \quad (4.24)$$

and

$$\int_{t=0}^{\infty} u'_i \, dt = 0, \quad \int_{t=0}^{\infty} p' \, dt = 0$$

where u_i , p are the mean values of the flow field and u'_i and p' are the fluctuating ones.

Substituting equations (4.24) into (4.1), (4.2) and time averaging we obtain:

$$\frac{\partial}{\partial x_j} (\rho u_j) = 0 \quad (\text{continuity}) \quad (4.25)$$

$$\frac{\partial}{\partial x_j} (\rho u_i u_j) = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (\mu \bar{e}_{ij} - \rho \overline{u'_i u'_j}) \quad (4.26)$$

where the overbars imply time averaged quantities. The extra terms $-\overline{\rho u'_i u'_j}$, which have been introduced in equations (4.26), by the time averaging process, are called the "Reynolds stresses" and represent transport of momentum by turbulent motions.

By a similar process we obtain for the energy equations (4.3) in circumstances giving assumptions of low Mach and Eckert numbers.

$$\frac{\partial}{\partial x_j} (\rho u_j c) = \frac{\partial}{\partial x_j} \left(\mu \sigma \frac{\partial c}{\partial x_j} - \overline{\rho u'_j c'} \right) \quad (4.27)$$

where c' are the fluctuations of non-dimensionalised temperature.

4.23 The closure problem

The mean momentum equations for turbulent flows contain the six unknown Reynolds-stresses elements and, in order therefore to render the set of momentum equations solvable we have to connect the Reynolds stresses with the mean hydrodynamic properties of the flow field. There are many ways of relating the Reynolds stresses to the mean hydrodynamic quantities (a discipline that is called modelling of turbulence); either by algebraic expressions or by, more complicated (and hopefully more realistic) partial differential equations.

It is possible to derive transport equations for the Reynolds stresses themselves: this is done by multiplying equation (4.2) by U_k and then time averaging, Hinze (52). This, however, does not solve the problem since it introduces new unknowns of the form $\overline{\rho u'_i u'_k u'_j}$ which are triple correlations of the fluctuating components of velocity and they too have to be modelled either algebraically or by another set of transport equations.

The triple correlations it is assumed that are of small influence in practical flow situations and consequently there is usually no justification for devising transport equations for their rate of change. Thus it would seem that the turbulence model which would be most suitable in the present study would calculate up to the level of the double correlations of the fluctuating velocities (i.e. the Reynolds stresses).

4.2.4 The modelling of the Reynolds stresses

It can be argued that the pressure field created by the injection is sufficiently strong that the Reynolds-stress terms in (4.26) will have little direct effect on the mean velocity field in the immediate vicinity of the hole. Downstream of injection however, it is the turbulent stress field which causes the flow field to revert towards a two-dimensional form. For the relatively low injection rates of the present study, $\partial(\overline{\rho u_1' u_2'})/\partial x_2$ is the largest of the stress gradients, (just as in a two-dimensional flat-plate boundary layer), with $\partial(\overline{\rho u_1' u_3'})/\partial x_3$ (which acts to accelerate the slow moving fluid behind the hole) the next most important.

Now in wall-generated turbulent shear flows, transport effects are known to be of minor importance and the turbulence structure is essentially in equilibrium. The injection process will, of course, locally distort this equilibrium, however because of the low levels of injection, it appeared that the assumption of local equilibrium was a reasonable starting point in approximating the Reynolds-stresses - particularly in view of the considerable simplification it brings. Thus by discarding transport effects from the stress closures of Rodi (53) and Launder (54) the following simple formula for the turbulent stress field is obtained

$$\left(\frac{\overline{u_i' u_j'}}{k} - \frac{2}{3} \delta_{ij} \frac{\overline{u_1'^2}}{\partial u_1'} \right) = 0.27 \left(P_{ij} - \frac{2}{3} \delta_{ij} P \right) / \epsilon \quad (4.28)$$

where $k \equiv \frac{1}{2} \overline{u_1'^2}$, $\epsilon = \nu \frac{\partial u_1'}{\partial x_j}$, and the quantities P_{ij} and P are respectively the local generation rates of $\overline{u_i' u_j'}$ and k , given by:

$$P_{ij} \equiv - \left(\overline{u_i' u_k'} \frac{\partial u_j}{\partial x_k} + \overline{u_j' u_k'} \frac{\partial u_i}{\partial x_k} \right)$$

$$P \equiv - \overline{u_i' u_j'} \frac{\partial u_i}{\partial x_j}$$

On neglecting gradients with respect to x_1 the shear components of the Reynolds stress tensor emerge from (4.28) as:

$$\overline{u'_1 u'_2} = -0.27 \frac{k}{\epsilon} \left(\overline{u'^2_2} \frac{\partial u_1}{\partial x_2} + \overline{u'_3 u'_2} \frac{\partial u_1}{\partial x_3} \right) \quad (4.29)$$

$$\overline{u'_1 u'_3} = -0.27 \frac{k}{\epsilon} \left(\overline{u'^2_3} \frac{\partial u_1}{\partial x_2} + \overline{u'_3 u'_2} \frac{\partial u_1}{\partial x_3} \right) \quad (4.30)$$

$$\overline{u'_2 u'_3} = -0.27 \frac{k}{\epsilon} \left(\overline{u'^2_3} \frac{\partial u_2}{\partial x_3} + \overline{u'^2_2} \frac{\partial u_3}{\partial x_2} \right) \quad (4.31)$$

In the present study the velocity gradients in the $x_2 - x_3$ plane are at least an order of magnitude smaller than those involving the streamwise velocity u_1 . Equations (4.29) - (4.31) suggest therefore that $\overline{u'_2 u'_3}$ is at least an order of magnitude less than $\overline{u'_1 u'_2}$ and $\overline{u'_1 u'_3}$. In fact, as we discuss later, equation (4.39) tends to underestimate the anisotropy of the stress components like $\overline{u'_2 u'_3}$ in which direct generation is small. Nevertheless we proceed on the basis that $\overline{u'_2 u'_3}$ is sufficiently small that the second terms in (4.29) and (4.30) may be neglected; this is a useful simplification to the task of numerical solution of the equations (4.25) - (4.26).

Consequently the two main shear stresses are presented by:

$$-\overline{u'_1 u'_2} = v_2 \frac{\partial u_1}{\partial x_2}; \quad -\overline{u'_1 u'_3} = v_3 \frac{\partial u_1}{\partial x_3} \quad (4.32)$$

where $v_i = 0.27k \overline{u'^2_i} / \epsilon$ (no summation on index i) that is the shear stresses are proportional to the mean strain but the turbulent "viscosities" in the lateral and normal directions are proportional to the normal stresses in these directions.

4.2.5 The modelling of the turbulent heat fluxes

Launder (55) has developed the heat-transfer counterpart of equation (4.28) to study stably stratified shear layers. For the case of negligible buoyant influence the following expression for the heat fluxes is obtained:

$$-\overline{u'_1 c'} = 0.31 \frac{k}{\varepsilon} (\overline{u'_1 u'_k} \frac{\partial c}{\partial x_k} + \frac{1}{2} \overline{u'_k c'} \frac{\partial u_1}{\partial x_k}) \quad (4.33)$$

Again, since we limit attention to cases where the streamwise velocity is much larger than those in the x_2 and x_3 directions the second term on the right of (4.33) may be neglected in the equations, for the lateral and normal to the wall heat fluxes. We then have:

$$-\overline{u'_2 c'} = 0.31 \frac{k}{\varepsilon} \overline{u'^2_2} \frac{\partial c}{\partial x_2}; \quad -\overline{u'_3 c'} = 0.31 \frac{k}{\varepsilon} \overline{u'^2_3} \frac{\partial c}{\partial x_j} \quad (4.34)$$

thus if we denote by Γ_2 the turbulent diffusion coefficient in the direction normal to the wall (i.e. $\Gamma_2 \equiv -\overline{u'_2 c'}/(\partial c/\partial x_2)$) and define Γ_3 correspondingly it follows that

$$\Gamma_3/\Gamma_2 = \overline{u'^2_3}/\overline{u'^2_2} \quad (4.35)$$

thus for the diffusive transport of both momentum and heat, the present proposals suggest that for low injection rates the ratio of the turbulent diffusivities is directly proportional to that of the normal stresses in the respective directions. This should be contrasted with the usual practice of the kind $\Gamma_3 = \Gamma_2$, i.e. assuming isotropy.

4.2.6 Assessment of the turbulence model proposed for the present study

We know of no reported measurements of v_3/v_2 or Γ_3/Γ_2 in developing boundary layers, consequently we are not able to check the validity of the above theoretical arguments. However for fully developed pipe flow (for which $\overline{u'^2_3}/\overline{u'^2_2}$ varies in a similar manner to that in a boundary layer) an early paper by Black and Sparrow (57) concluded that Γ_3 had to be considerably greater than Γ_2 to explain the measured near-wall temperature profiles. More recently Quarmby and Quirk (58) (59) have reported extensive data on Γ_3/Γ_2 from measurements of asymmetric gaseous diffusion in a pipe. In a recent survey of heat and mass transfer data, reported by Launder (60)

we concluded that equation (4.32) provides a reasonable fit to these data considering their scatter; the diffusivities, however, appear to display somewhat greater anisotropy than the normal stresses. In the present work in order to simplify the numerical work involved we adopted the following fit to the Quarmby-Quirk data;

$$\begin{aligned} \frac{v_3}{v_2} &= \frac{\Gamma_3}{\Gamma_2} = 1 + 3.5 (1-x_2/\Delta) , & x_2 < \Delta \\ \frac{v_3}{v_2} &= \frac{\Gamma_3}{\Gamma_2} = 1.0 , & x_2 > \Delta \end{aligned} \quad (4.36)$$

where Δ is the local height above the surface at which $u_1/u_{1\infty}$ reaches 0.95.

At present in addition to k and ϵ the quantity $\overline{u_2'^2}$ appears in equations (4.32) and (4.34) for v_2 and Γ_2 . Partly to simplify further the numerical solution we replace $\overline{u_2'^2}$ by $k/3$ which provides a good mean fit over the inner 50% of the boundary layer; thus we obtain:

$$v_2 = 0.09 k^2/\epsilon ; \quad \Gamma_2 = 0.10 k^2/\epsilon \quad (4.37)$$

The other reason for eliminating $\overline{u_2'^2}$ as indicated is that the resultant formula for v_2 and Γ_2 are then in agreement with those adopted in the k - ϵ turbulence model - a closure that has been used with considerable success by numerous workers in predicting two-dimensional flows (see, for example, Launder and Spalding (61) for a review). As in that model, the turbulence energy and its rate of dissipation are obtained from the solution of the following pair of transport equations (here expressed in the ξ, n, ζ coordinate system).

$$\begin{aligned} \frac{\partial}{\partial \xi} (\rho u k) + \frac{1}{\delta} \left(\frac{\partial}{\partial \eta} (\rho v k) - \eta \left(\frac{d\delta}{dx} \right) \frac{\partial}{\partial \xi} (\rho u k) \right) + \frac{\partial}{\partial \xi} (\rho w k) = \\ \frac{1}{\delta} \frac{\partial}{\partial \eta} \left(\left(\mu + 0.09 \rho \frac{k^2}{\epsilon} \right) \frac{1}{\delta} \frac{\partial k}{\partial \eta} \right) + \frac{\partial}{\partial \zeta} \left(\left(\mu + 0.09 \rho \frac{v_3}{v_2} \frac{k^2}{\epsilon} \right) \frac{\partial k}{\partial \zeta} \right) + \rho (P - \epsilon) \end{aligned} \quad (4.38)$$

$$\begin{aligned} \frac{\partial}{\partial \xi} (\rho u \epsilon) + \frac{1}{\delta} \left(\frac{\partial}{\partial \eta} (\rho v \epsilon) - \eta \left(\frac{d\delta}{dx} \right) \frac{\partial}{\partial \xi} (\rho u \epsilon) \right) + \frac{\partial}{\partial \zeta} (\rho w \epsilon) = \\ \frac{1}{\delta} \frac{\partial}{\partial \eta} \left(\left(\mu + 0.07 \rho \frac{k^2}{\epsilon} \right) \frac{1}{\delta} \frac{\partial \epsilon}{\partial \eta} \right) + \frac{\partial}{\partial \zeta} \left(\left(\mu + 0.07 \rho \frac{v_3}{v_2} \frac{k^2}{\epsilon} \right) \frac{\partial \epsilon}{\partial \zeta} \right) + \\ 1.44 \rho \frac{P \epsilon}{k} - 1.9 \rho \frac{\epsilon^2}{k} \end{aligned} \quad (4.39)$$

4.2.7 The inlet and boundary conditions for K, ϵ

No attempt was made to extend the numerical computations into the region immediately adjacent to the wall where viscous effects are important. Although a version of the k- ϵ model has been devised for this region by Jones and Launder (62), it has been tested only for two-dimensional boundary layers, the fine mesh required to resolve the immediate near-wall region adequately would have increased the present computing times by at least a factor of three - an increase which we felt could not be justified.

The study therefore makes use of "wall functions" to obviate the need to place a great number of grid nodes in the near wall region. Details of the treatment of this region will be deferred until the solution procedure is described; as inlet conditions for K and ϵ the following relations were used, applicable for 2-dimensional boundary layer type of flows :
(for more details see ref. (61))

$$\begin{aligned} K = \tau / \rho \sqrt{c_D} = l^2 (du/dy)^2 / \sqrt{c_D} \\ \epsilon = K^{1.5} / L = c_D^{.75} K^{1.5} / l \end{aligned}$$

where l is Prandtl's mixing length, $l = 0.41 y$ ($l_{\max} = c_D \delta$)

and L is the turbulence length scale.

4.2.8 Summary

Summarised below are the main points made in this section on the modelling of turbulence.

- (i) The time averaging procedure of the Navier Stokes equation and energy equation introduces into the set of equation additional unknowns, in the form of the Reynolds stresses and the turbulent heat fluxes.
- (ii) The stress closure of Rodi (53) and Launder (54) and Launder's (55) closure for the heat transfer counterpart of the stresses are used to model the Reynolds stresses and the turbulent heat fluxes, with simplifications allowed by the particular circumstances of the DHC flow.
- (iii) It has been shown that the diffusivities of momentum and heat in the lateral direction are much larger than in the normal direction and, in particular, that they are proportional to the corresponding normal stresses.
- (iv) The turbulence energy and scale are obtained from the $k-\epsilon$ equations appropriately modified to account for the non-isotropic character of the diffusivities.
- (v) The set of equations of the model here employed comprises equations,
(4.38) (4.39) (4.36)

4.3 The numerical solution of the equations

4.3.1 Introduction

The mathematical formulation of the problem showed that the conservation equations governing the flow field are difficult to solve analytically due to their complexity. With the advent of digital computers however, it has been possible to solve the three-dimensional Navier-Stokes equations by the application of numerical integration techniques.

Since 1967 a number of numerical procedures have been reported for solving the three dimensional boundary layer form of the Navier-Stokes equations.

The first step in the development of a numerical procedure for solving the differential equations is to superimpose over the flow domain a grid distribution and to discretize the differential equations on all the grid points of the flow field and to obtain "equivalent" algebraic expressions called finite difference equations (fde). The accuracy of the set of the finite difference equations which approximates the partial differential equations is very much dependent on the formulation of the fde's and on the number of the grid nodes which map the flow field.

Basically three different approaches have been followed to obtaining finite difference equations from the differential equations;

- (i) by Taylor-series expansion and the Truncation of the series;
- (ii) integration over finite elements employing some variational principle;
- (iii) by integration of the differential equations over small control volumes surrounding each grid node.

The second method, that of applying a variational principle, is associated with the finite element technique and a critical examination of the resultant finite difference equations using methods (ii) and (iii) shows refs (63) and (64), that in most cases the resultant equations are the same. As regards the method of expansion in Taylor series, the method is less general than (ii) and (iii) because apart from offering less physical insight in the

derivation of the fde's the reciprocity requirement for the fluxes at locations midway between grid points usually leads to the central difference formulation which is inaccurate representation of the fluxes at high Peclet numbers.

Patankar (65) in 1971 reviewed the then available calculation procedures for 3D parabolic flows and he concluded that most of them have assumed that

$$\begin{aligned} \text{(i)} \quad \frac{\partial p}{\partial y} &= 0, \quad \frac{\partial p}{\partial z} = 0 \text{ and} \\ \text{(ii)} \quad \tau_{zx} &= 0 \end{aligned} \tag{4.40}$$

and some others neglect the terms

$$\text{(iii)} \quad v \frac{\partial v}{\partial y} \quad \text{and} \quad v \frac{\partial w}{\partial y} \tag{4.41}$$

from the momentum equations and thus are restricted to small cross-stream velocities. However in the present flow situation none of the above assumption can be regarded as valid since the cross stream velocities are not small (because of injection) and the shear stress τ_{zx} is of the same order of importance as τ_{yx} . Consequently it was decided to employ the numerical solution procedure which was developed by Patankar and Spalding (66) as the most general of all the available procedures without being restricted by any other assumptions except those of the boundary layer type mentioned in section 4.1.3.

Before we proceed in the presentation of the numerical scheme it is helpful to note that all the conservation equations may be cast into the same form

$$\begin{aligned} \frac{\partial}{\partial \zeta} (\rho u \phi) + \frac{1}{\delta} \left(\frac{\partial}{\partial \eta} (\rho v \phi) - \eta \frac{d\delta}{dx} \frac{\partial}{\partial \eta} (\rho u \phi) \right) + \frac{\partial}{\partial \zeta} (\rho w \phi) = \\ \frac{1}{\delta} \frac{\partial}{\partial \eta} \left(\frac{1}{\delta} \Gamma_{n,\phi} \frac{\partial \phi}{\partial \eta} \right) + \frac{\partial}{\partial \zeta} \left(\Gamma_{\zeta,\phi} \frac{\partial \phi}{\partial \zeta} \right) + S_{\phi} \end{aligned} \tag{4.42}$$

where ϕ stands for any of the dependent variables, u, v, w, k, ϵ, T . S_{ϕ} is

a "source" term which for each conservation equation takes the value shown in table 4; $\Gamma_{n,\phi}$ or $\Gamma_{\zeta,\phi}$ is the turbulent effective diffusion coefficient for ϕ also defined on table 4 (note that the numerical procedure is not restricted by the particular form of $\Gamma_{,\phi}$)

4.3.2 The procedure for discretizing the differential equations

(1) Outline of the procedure

Here the procedure by which the partial differential equations are discretized to give the finite difference equations is presented. The analysis applies both to parabolic and semi-elliptic procedures while their differences in the solution procedure itself is outlined at a latter stage. The sequence of discretization of the equations is as follows:

- (a) The domain of interest is divided into a set of non-overlapping sub-domains by drawing surfaces of constant ξ, η and ζ , figure 57d. This division is arbitrary and the greater the number of sub-domains (called cells or control volumes) the more accurate is the representation of the differential equations by the fde's.
- (b) The locations of the dependent variables differs according to the quantity in questions. In the "SIMPLE" solution algorithm of Patankar and Spalding (66) ("SIMPLE" stands for Semi - Implicit - Method - for Pressure - Linked - Equations) the quantities P, T, K and ϵ are calculated at the intersection of the grid lines (grid nodes) while the velocity components u, v, w are located at locations halfway between the grid nodes. Chorin and Harlow (67) invented this arrangement. It is used because the velocities depend on pressure differences and not on pressures. The location for each dependent variable is shown on figure 57d.
- (c) Then the partial differential equations are integrated with the aid of approximations over the appropriate control volumes. There are the following points to be noticed:

equation	Γ_Φ	Source term S_Φ
u	$\Gamma_{\eta,u} = \mu + 0.09 \rho k^2 / \epsilon$ $\Gamma_{\xi,u} = \mu + 0.09 r^* \rho k^2 / \epsilon$	$-\frac{\partial P}{\partial \xi} + \frac{1}{\delta} \eta \frac{d\delta}{dx} \frac{\partial P}{\partial \eta}$
v	$\Gamma_{\eta,v}$ as above (η, u) $\Gamma_{\xi,v}$ as above (ξ, u)	$-\frac{1}{\delta} \frac{\partial P}{\partial \eta} + \frac{1}{\delta^2} \frac{\partial}{\partial \eta} \left\{ \Gamma_{\eta,v} \frac{\partial v}{\partial \eta} \right\} + \frac{1}{\delta} \frac{\partial}{\partial \xi} \left\{ \Gamma_{\eta,v} \frac{\partial w}{\partial \eta} \right\}$
w	$\Gamma_{\eta,w}$ as above $\Gamma_{\xi,w}$ as above	$-\frac{\partial P}{\partial \xi} + \frac{1}{\delta} \frac{\partial}{\partial \eta} \left\{ \Gamma_{\xi,w} \frac{\partial w}{\partial \xi} \right\} + \frac{\partial}{\partial \xi} \left\{ \Gamma_{\xi,w} \frac{\partial w}{\partial \xi} \right\}$
K	$\Gamma_{\eta,k}$ as above $\Gamma_{\xi,k}$ as above	$0.09 \rho k^2 / \epsilon G - \rho \epsilon$ where $G = \left\{ 2 \left\{ \frac{1}{\delta^2} \left\{ \frac{\partial v}{\partial \eta} \right\}^2 + r \left\{ \frac{\partial w}{\partial \xi} \right\}^2 \right\} + \frac{1}{\delta^2} \left\{ \frac{\partial u}{\partial \eta} \right\}^2 + r \left\{ \frac{\partial u}{\partial \xi} \right\}^2 + \left\{ \frac{1}{\delta} \frac{\partial w}{\partial \eta} + \frac{\partial v}{\partial \xi} \right\}^2 \right\}$
ϵ	$\Gamma_{\eta,\epsilon} = \mu + 0.07 \rho k^2 / \epsilon$ $\Gamma_{\xi,\epsilon} = \mu + 0.07 r \rho k^2 / \epsilon$	$1.44 \rho k G - 1.9 \rho \epsilon^2 / k$
T	$\Gamma_{\eta,T} = \mu + 0.1 \rho k^2 / \epsilon$ $\Gamma_{\xi,T} = \mu + 0.1 r \rho k^2 / \epsilon$	0

Table 4 : S_Φ and Γ_Φ for each dependent variable Φ

*r equals v_3/v_2 relation (4.36)

(i) The location of the velocities is such that they lie at the interfaces of the control volume for the continuity equation, where they are needed for the calculation of the mass fluxes.

(ii) The interface densities are evaluated as arithmetic means of the neighbour nodal values; e.g. $\rho_e = \frac{1}{2} (\rho_E + \rho_P)$

(iii) The equations can be integrated once over the control volume and the resulting surface integrals are approximated by the use of the mean value theorem.

(2) The derivation of the finite difference equations

The integration of the differential equations follows a sequence suggested by Gosman (68).

(a) The finite difference equation for the continuity

The partial differential equation (p.d.e) (4.20) is integrated over the control volume shown in figure 57e. The result is

$$\begin{aligned} \int_V (\text{p.d.e}) \, dv &= (\rho u)_d (\delta \Delta n \Delta \zeta)_d - (\rho u)_u (\delta \Delta n \Delta \zeta)_u + \\ &\left(\left((\rho v)_n - \left(n \frac{d\delta}{dx} \right)_n (\rho u)_n \right) (\Delta \xi \Delta \zeta)_n - \left((\rho v)_s - \left(n \frac{d\delta}{dx} \right)_s (\rho u)_s \right) (\Delta \xi \Delta \zeta)_s \right. \\ &\left. + (\rho w)_e (\delta \Delta n \Delta \xi)_e - (\rho w)_w (\delta \Delta n \Delta \xi)_w \right) = 0 \end{aligned} \quad (4.43)$$

(b) The finite difference equation for T, k and ε

The partial differential equation for temperature, turbulent kinetic energy and its dissipation rate, equation (4.42) is integrated over a control volume surrounding the grid node under consideration, figure 57e.

(i) Integration of the fluxes

The combined flux in, say, η direction comprising both convection and diffusion flux is integrated as follows:

$$\int_{V,\eta} \left\{ \frac{1}{\delta} \left(\frac{\partial}{\partial \eta} (\rho v \phi) - \eta \frac{d\delta}{dx} \frac{\partial}{\partial \eta} (\rho u \phi) \right) - \frac{\partial}{\partial \eta} \left(\Gamma_{\eta,\phi} \frac{1}{\delta} \frac{\partial \phi}{\partial \eta} \right) \right\} dV =$$

$$\left\{ ((\rho v \phi)_n - (\eta \frac{d\delta}{dx})_n (\rho u \phi)_n) - (\Gamma_{\eta,\phi} \frac{1}{\delta} \frac{\partial \phi}{\partial \eta})_n \right\} (d\xi d\zeta)_n$$

$$- \left\{ (\rho v \phi)_s - (\eta \frac{d\delta}{dx})_s (\rho u \phi)_s - (\Gamma_{\eta,\phi} \frac{1}{\delta} \frac{\partial \phi}{\partial \eta})_s \right\} (d\xi d\zeta)_s$$

With the basic assumptions stated earlier in the outline of the procedure, the above equation is transformed to

$$\int_{V,\eta} (\text{flux}) dV = (\Delta \xi \Delta \zeta)_n \left((\rho v) - (\eta \frac{d\delta}{dx})_n (\rho u) \right)_n (f_n \phi_P + (1-f_n) \phi_N)$$

$$- (\Delta \xi \Delta \zeta)_s \left((\rho v) - (\eta \frac{d\delta}{dx})_s (\rho u) \right)_s (f_s \phi_S + (1-f_s) \phi_P) \quad (4.44)$$

where f_n and f_s are weighting factors, to be calculated in the next paragraph.

Similarly the finite difference approximation for the combined fluxes in the ζ and ξ direction* are expressions (4.45) and (4.46)

$$\int_{V,\zeta} (\text{flux}) dV = (\Delta \xi \Delta \eta \delta) (\rho w)_e (f_e \phi_P + (1-f_e) \phi_E) -$$

$$(\Delta \xi \delta \Delta \eta)_w (\rho w)_w (f_w \phi_W + (1-f_w) \phi_P) \quad (4.45)$$

$$\int_{V,\xi} (\text{flux}) dV = (\Delta \zeta \delta \Delta \eta)_d (\rho u)_d (f_d \phi_P + (1-f_d) \phi_D) -$$

$$(\Delta \zeta \delta \Delta \eta)_u (\rho u)_u (f_u \phi_U + (1-f_u) \phi_P) \quad (4.46)$$

(ii) Calculation of the weighting factor f_i

In deriving expression (4.44) for the combined flux in the y direction the value of the dependent variable ϕ_n at the north face n of the control volume was approximated as a weighting mean of the values of ϕ at the N and P grid nodes

* The formulation of (4.46) is general, as if the diffusion fluxes in streamwise direction were not zero.

$$\phi_n = g_n \phi_N + (1-g_n) \phi_P \quad (4.47)$$

The weighting factor g_n is calculated on the assumption that the variation of the dependent variable ϕ along the normal to the face n (along the PN line, fig.57†) is given by the solution of the one-dimensional counterpart of the combined flux, i.e.

$$\frac{1}{\delta} ((pv) - (\eta \frac{d\delta}{dx}) (\rho u))_n \frac{\partial \phi}{\partial \eta} - \frac{1}{\delta} \frac{\partial}{\partial \eta} (\Gamma_{n,\phi} \frac{1}{\delta} \frac{\partial \phi}{\partial \eta}) = 0 \quad (4.48)$$

The solution of the above equation is given by

$$\phi = g \phi_N + (1-g) \phi_P \quad (4.49)$$

where $g = \left(\exp (Pe_n (\eta - \eta_P) / (\eta_N - \eta_P)) - 1 \right) / \left(\exp (Pe_n) - 1 \right)$ (4.50)

and the Peclet number Pe_n is defined as

$$Pe_n = \rho_n (v - \eta \frac{d\delta}{dx})_n \delta (\eta_N - \eta_P) / \Gamma_{n,\phi} \quad (4.51)$$

From the above analysis, the weighting factor g_n , used in equation (4.47) is calculated as:

$$g_n = \left(\exp (Pe_n) - 1 \right) / \left(\exp (Pe_n) - 1 \right) \quad (4.52)$$

Similarly we obtain the values of the weighting factor g_e, g_d etc.

Use of expression (4.47) with the approximation that

$$\Gamma_{n,\phi} \frac{\partial \phi}{\partial \eta} = \Gamma_{n,\phi} \frac{\phi_N - \phi_P}{\delta (\eta_N - \eta_P)} \quad (4.53)$$

leads to expression (4.44) where

$$f_n = 1 - g_n + \frac{1}{Pe_n} \quad (4.54)$$

$$f_s = 1 - g_s + \frac{1}{Pe_s} \quad (4.55)$$

$$f_d = 1 - g_d + \frac{1}{Pe_d} \quad (4.56)$$

(iii) The upwind formulation of the streamwise flux

The employed, in the present study, set of equations is parabolic, hence it is assumed that the diffusion fluxes in the streamwise direction are zero, consequently it is taken that

$$\Gamma_{\xi, \phi} = 0 \quad (4.57)$$

Mathematically equation (4.56) has the limit of 1. as $\Gamma_{\xi, \phi}$ tends to zero. Hence for parabolic type of flows f_d equals 1. and consequently the previous analysis leads to

$$\phi_d = \phi_p \quad \text{and} \quad \phi_u = \phi_u \quad (4.58)$$

This approximation of ϕ_d and ϕ_u by their "upwind" values is also called the upwind difference formula and, as seen this is a particular case of the general formulation (4.44) for high local Peclet numbers.

(iv) The hybrid flux expression

Because exponentials are expensive to compute the exact expression for the weighting factor f is approximated, with little loss of accuracy, by the following expression ref (69).

$$f = \frac{1}{2} \left(1 + \frac{2}{Pe} \right) \quad \text{if } |Pe| < 2 \quad (4.59)$$

$$f = 1 \quad \text{if } Pe > 2 \quad (4.60)$$

$$f = 0 \quad \text{if } Pe < -2 \quad (4.61)$$

Expressions (4.60) and (4.61) evidently are associated with the "upwind" formulation of the fluxes for high Peclet numbers, $Pe > 2$, while expression (4.59) leads to the central difference one.

(v) Integration of the source expression

Generally the integral of the source term is expressed in linearised form as

$$\int_V s_\phi dV = b \phi_P + c$$

where b and c may be functions of ϕ .

(vi) Assmbeily of the finite-difference equation

Using the previously derived expressions for the fluxes and source we obtain, with the use of the continuity equation, the following finite difference equation for the ϕ dependent variable

$$(A_P^\phi - b^\phi) \phi_P = A_E^\phi \phi_E + A_W^\phi \phi_W + A_N^\phi \phi_N + A_S^\phi \phi_S + A_U^\phi \phi_U + C^\phi \quad (4.62)$$

where

$$A_E^\phi = (\rho w)_e (\delta \Delta \eta \Delta \xi)_e \{f_e - 1\}$$

$$A_N^\phi = \left((\rho v)_n - \left(\eta \frac{d\delta}{dx} \right)_n (\rho u)_n \right) (\Delta \xi \Delta \zeta)_n \{f_n - 1\} \quad (4.63)$$

$$A_U^\phi = (\rho u)_u (\delta \Delta \eta \Delta \zeta)_u, \text{ etc and}$$

$$A_P^\phi = A_E^\phi + A_W^\phi + A_N^\phi + A_S^\phi + A_U^\phi \quad (4.64)$$

(c) The finite difference equations for u,v and w

Following a similar path to that outlined in some detail earlier we obtain the fde's for the u,v and w variables by integrating the corresponding momentum equations on control volumes shown in figures 57d, 57e. The resulting fde's are similar to (4.62)

$$(A_P^u - b^u) u_P = A_E^u u_E + A_W^u u_W + A_N^u u_N + A_S^u u_S + A_U^u u_U +$$

$$(\delta \Delta \eta \Delta \zeta)_d (P_P - P_D) + C^u \quad (4.65)$$

$$(A_P^v - b^v) v_P = A_E^v v_E + A_W^v v_W + A_N^v v_N + A_S^v v_S + A_U^v v_U + (\Delta \xi \Delta \zeta)_S (P_S - P_P) + C^v \quad (4.66)$$

$$(A_P^w - b^w) w_P = A_E^w w_E + A_W^w w_W + A_N^w w_N + A_S^w w_S + A_U^w w_U + (\Delta \xi \delta \Delta \eta)_W (P_W - P_P) + C^w \quad (4.67)$$

where in all cases

$$A_P^\phi = A_E^\phi + A_W^\phi + A_N^\phi + A_S^\phi + A_U^\phi$$

4.3.3 The "SIMPLE" algorithm

The "SIMPLE" algorithm of Patankar and Spalding (66) is a guess-and-correct procedure along the lines of the earlier methods of Chorin and Amsdent-Harlow. The main idea is that if the pressure field were known the finite difference equations for the momentum components could be solved; consequently a guess of the pressure field is made, say p^* (for parabolic flows the pressure field is guessed only on the cross-stream planes in question, while for semi-elliptic flows p^* is estimated for the whole flow field). With the guessed pressure field p^* the fde (4.65) - (4.67) are solved. The velocity field thus obtained, u^* , v^* and w^* , based on p^* does not generally satisfy the continuity equation (4.43) and the mass imbalance thus calculated for each control volume

$$\begin{aligned} m_P^* = & (\rho u^*)_d (\delta \Delta \eta \Delta \zeta)_d - (\rho u^*)_u (\delta \Delta \eta \Delta \zeta)_u + ((\rho v^*)_n - (\eta \frac{d\delta}{dx})_n (\rho u^*)_n) (\Delta \zeta \Delta \xi)_n \\ & - ((\rho v^*)_s - (\eta \frac{d\delta}{dx})_s (\rho u^*)_s) (\Delta \xi \Delta \zeta)_s + (\rho w^*)_e (\delta \Delta \eta \Delta \xi)_e - \\ & (\rho w^*)_w (\delta \Delta \eta \Delta \xi)_w \end{aligned} \quad (4.68)$$

is used to derive corrections of the guessed pressure field, p' , so these mass imbalances are reduced to zero.

Another basic characteristic of the "SIMPLE" algorithm is the use of a linearised momentum equation for the velocities. If we assume that u velocity is a function of pressure only

$$u = u(p)$$

then we derive

$$u \sim u^* + \frac{\partial u}{\partial (\Delta p^*)} \Delta p^* \quad (4.69)$$

where Δp^* is the driving pressure difference.

Now, if we also take that the guessed pressure field is corrected by local amounts p' , then the above relation gives

$$\begin{aligned} u &= u^* + \frac{\partial u}{\partial (p_P^* - p_D^*)} (p_P' - p_D') \quad \text{and similarly} \\ v &= v^* + \frac{\partial v}{\partial (p_P^* - p_S^*)} (p_P' - p_S') \\ w &= w^* + \frac{\partial w}{\partial (p_P^* - p_W^*)} (p_P' - p_W') \end{aligned} \quad (4.70)$$

The coefficients $\frac{\partial u}{\partial (p_P^* - p_D^*)}$, etc. in equation (4.70) are obtained from the finite difference momentum equations, (4.65)-(4.67). Thus:

$$\begin{aligned} DU &= \frac{\partial u}{\partial (p_P^* - p_D^*)} = \frac{(\delta \Delta \eta \Delta \zeta)_d}{(A_P^u - b^u)} \\ DV &= \frac{\partial v}{\partial (p_P^* - p_S^*)} = \frac{(\Delta \xi \Delta \zeta)_s}{(A_P^v - b^v)} \\ DW &= \frac{\partial w}{\partial (p_P^* - p_W^*)} = \frac{(\Delta \xi \delta \Delta \eta)_w}{(A_P^w - b^w)} \end{aligned} \quad (4.71)$$

Finally we get the following relations:

$$\begin{aligned}
 u &= u^* - D\mathbf{U}(p'_D - p'_P) \\
 v &= v^* - D\mathbf{V}(p'_P - p'_S) \\
 w &= w^* - D\mathbf{W}(p'_P - p'_W)
 \end{aligned}
 \tag{4.72}$$

The above u, v, w velocities, equation (4.72), are required to satisfy the continuity equation (4.43). Substitution of (4.72) into (4.43) and after rearrangement result in the following pressure correction equation;

$$A_{P-P}^P p'_P = A_{E-E}^P p'_E + A_{W-W}^P p'_W + A_{N-N}^P p'_N + A_{S-S}^P p'_S + A_{D-D}^P p'_D + m_p^* \tag{4.73}$$

where

$$\begin{aligned}
 A_E^P &= (\Delta\xi\delta\Delta\eta)_e DW \\
 A_D^P &= (\delta\Delta\eta\Delta\xi)_d DU, \text{ etc.}
 \end{aligned}$$

m_p^* taken from equation (4.68) and

$$A_P^P = A_E^P + A_W^P + A_N^P + A_S^P + A_D^P$$

4.3.4 The treatment of the pressure field

In the mathematical foundation of the problem it was stated that the set of equations (4.20)-(4.22) was regarded as parabolic or semi-elliptic set depending on the nature of the Poisson equation for the pressure. Below we outline the differences in the calculation procedure for parabolic and semi-elliptic flows.

(1) Parabolic flows

The treatment of the pressure field is particular in the Patankar-Spalding procedure (66). The fact that the Poisson equation for the pressure is elliptic only on cross-stream planes and parabolic in the streamwise direction allows in principle a marching integration technique to be applied for the solution of the governing differential equations. The transformation of the Poisson equation to be elliptic only on cross-stream planes is achieved by decoupling the streamwise pressure gradient from those on cross-

stream planes. The value of the streamwise pressure gradient is:

- (a) For unconfined flows, taken to be equal to the streamwise pressure gradient at the free surface, which is specified;

$$\frac{\partial p}{\partial \xi} = \left(\frac{dp}{d\xi} \right)_{\text{mean}} = \left(\frac{\partial p}{\partial \xi} \right)_{\infty};$$

- (b) For confined flows, is calculated from the requirement of satisfaction of over-all mass balance within each cross-stream section. Consequently, for parabolic flows, the term expressing the streamwise pressure gradient in the u momentum fde, (4.65), $(\delta \Delta \eta \Delta \zeta)_d (p_D - p_P)$ is known and equal to $-(\Delta \xi \delta \Delta \eta \Delta \zeta)_d \left(\frac{dp}{d\xi} \right)_{\infty}$

(2) Semi-elliptic flows

If the decoupling of the streamwise pressure gradient is not justified, then the entire 3D pressure field has to be computed simultaneously with the other unknown variables.

4.3.5 The pressure correction equation for parabolic and semi-elliptic flows

(1) Parabolic flows

For parabolic type external flows the streamwise pressure gradient is known. Consequently in the solution procedure we do not make guess of any streamwise pressure gradient, hence relation (4.71) implies that

$$DU = 0 \quad (4.74)$$

Consequently in the pressure correction equation, (4.73) the coefficient A_D^P is equal to zero.

$$A_D^P = 0 \quad (4.75)$$

(2) Semi-elliptic flows

For semi-elliptic flows the guessed pressure field p^* applies in all three directions and consequently in this case;

$$DU \neq 0 \quad \text{and} \quad A_D^p \neq 0 \quad (4.76)$$

However in the solution of the pressure correction equation not only the values of p' on the cross-stream plane (p'_p, p'_E, p'_W, p'_S) are unknown but also those on the downstream plane, p'_D . In order to adopt a marching integration technique we assume that p'_D is zero which, evidently will eventually be correct as the guessed pressure field converges towards the correct pressure field; the question is whether it is satisfactory practice to invoke before convergence.

The above proposals for semi-elliptic flows that

$$A_D^p \neq 0 \quad \text{and}$$

$$p'_D = 0$$

were originally suggested by Spalding, and were incorporated in Pratap's work (70). However, if we employ this practice assuming p'_D zero, then relation (4.72) becomes

$$u = u^* + DU_p'$$

and during the course of the present work it was found that this relation sometimes prohibits convergence since it contradicts with the physical reality. The reason is that it implies, for a situation where the pressure field is building up (e.g. upstream of the injection hole) the the fluid accelerates ($u > u^*$), since $p'_D > 0$ and $DU > 0$.

This contradiction is a consequence of assuming that $p'_D = 0$, whereas physical reality suggests that

$$p'_D > p'_P > 0.$$

In the present study since p'_D is unknown we retained the $p'_D = 0$ assumption but did not subsequently use equation (4.72) to correct the streamwise velocity: the correction was effectively applied when the newly-adjusted pressures were used in the next iteration on u velocity.

4.3.6 Summary of the SIMPLE algorithm

Below it is presented in summary the sequence in which the finite difference equations (4.65) - (4.67), (4.62) are solved, using the "SIMPLE" algorithm for parabolic flows.

- (1) A pressure field is guessed, p^* on cross-stream planes.
- (2) The coefficients of the finite-difference equations for the u momentum are calculated, based on upstream or inlet values of the velocity field, and the finite difference equation for u velocity (4.65) is solved by the tridiagonal matrix algorithm (TDMA).
- (3) The above step is repeated for the lateral, v and w, velocities, equations (4.66) and (4.67).
- (4) The mass imbalances are calculated for each cell using the calculated u^* , v^* , w^* velocities (4.68). The coefficients of the pressure correction equation are calculated and the equation (4.73) is solved.
- (5) The lateral velocities are now corrected using the calculated pressure corrections and relations (4.72) and a new pressure field is obtained ($p = p^* + p'$).
- (6) Then the finite difference equations for k, ϵ or T (4.62), are solved (the coefficients of fde are calculated using the newly obtained velocity field).

(7) Thus the velocity and temperature field have been obtained on the cross-stream plane under consideration and then the procedure marches downstream to the next plane for integration. This cycle (1) - (7) is repeated until the whole flow field is scanned.

For semi-elliptic flows the guessed pressure field in step (1) is for the whole flow field, $p^*=p^*(\xi, \eta, \zeta)$ and the sequence of calculations follows exactly the steps (1) - (7). However, when the whole flow field is scanned the sequence of calculations is repeated using as new guess of the pressure field the one calculated from the previous solution of the flow field, until a converged solution of the pressure field is achieved.

(4.3.7) The wall numerical boundary conditions.

In the near wall region there are very steep gradients of the dependent variables. Consequently for an adequate representation of this region an excessive amount of grid nodes would be required with result to increase the computer storage requirement as well as the computer time. Instead the practice of representing this region by the so called wall functions (71) has been adopted. The feature of the method is to locate all the finite-difference grid nodes in the fully turbulent region. Thus the near wall point P, figure 57g, is located sufficiently far from the wall for the local turbulent Reynolds number $(\rho k_P^2 y_P / \mu)$ to be much greater than unity.

It is known that in the near wall region the velocity profiles as well as the structure of turbulence exhibit some universal characteristics, mainly that:

(1) The velocity profile obeys the logarithm law (in the region $y^+ > 35$)

$$\frac{u}{u_\tau} = \frac{1}{0.41} \ln(9 u_\tau y_P \rho / \mu) \quad (4.77)$$

where u_τ is the friction velocity.

- (2) Generation of turbulence is equal its dissipation rate and,
- (3) this region is a constant shear stress region.

Relation (4.77) is applied as boundary condition for the velocity components parallel to and adjacent to wall boundaries of the flow.

Assumptions (2) and (3) with the employed form of the k equation lead to relation (4.78)

$$u_\tau = 0.09^{1/2} k_p^{1/2} \quad (4.78)$$

for the friction velocity u_τ which is used in relation (4.77) where k_p is the turbulent kinetic energy at location P.

The rate of dissipation of kinetic energy near the wall is fixed by the requirement that the length scale varies linearly with the distance from the wall; the expression for ϵ_p is

$$\epsilon_p = 0.09^{0.75} k_p^{1.5} / 0.41 y_p \quad (4.79)$$

Finally the quantity k_p is calculated from the limiting form of its conservation equation assuming that the turbulent kinetic energy diffusion to the wall being equal to zero,

$$\left(\frac{\partial k}{\partial y} \right)_{\text{wall}} = 0 \quad (4.80)$$

4.3.8 The relative merits of the parabolic and semi-elliptic procedures

The basic difference between the parabolic and semi-elliptic procedures is that the latter iterates in the streamwise direction, consequently repeating the parabolic procedure many times until the pressure field has sufficiently converged (as indicated by the smallness of the mass sources). Hence the semi-elliptic procedure is more expensive in computer time than the parabolic one. However the flow field solved by a semi-elliptic

procedure can not be handled by a parabolic but only by a fully elliptic one. Such a procedure, however, which in principle would operate in the same way as a semi-elliptic one, would require 3 dimensional storage for all the dependent variables $(u,v,w,p,k,\epsilon,T,\rho)$ and not merely for the pressure as the semi-elliptic requires. Hence it is evident the considerable economy of the semi-elliptic procedure as regards computer storage, a factor which is very short in the today computers.

The question arises how much more expensive in computer time is the semi-elliptic procedure in comparison with the parabolic one. Calculations performed in the present study indicate that:

- (1) For the present problem with an employed grid 20×20 and 50 axial steps a fully converged solution of the semi-elliptic procedure required around 70 iterations (in the streamwise direction).
- (2) The forward integration step allowed by a semi-elliptic procedure is 5-10 times greater than the corresponding step required by a parabolic procedure (due to the iterative character of the former).

As a result of the above findings it is concluded that with the employed grids the semi-elliptic procedure is about 10 times more expensive than the parabolic procedure and requires less than quarter of the computer storage needed for a fully elliptic treatment.

4.3.9 Some other aspects of the numerical procedure

The topics of convergence of the solution procedure, stability and details of the triangular matrix algorithm are not presented here for reasons of brevity of the thesis. They can be found in the works of Tatchell (72) and Pratap (70).

Some particular numerical aspects related to the treatment of the reverse flow region, the free boundary etc are discussed in Appendix B. Tests for grid dependency are reported in next chapter. The final grid distribution employed was 20×20 on the cross-stream planes, with

50 axial steps (100 steps for the double row case). The time involved was 0.03 seconds per iteration - per equation for each cross-stream plane on the IBM 360 computer.

CHAPTER 5

VALIDATION OF THE NUMERICAL PROCEDURE

5.1 Introductory remarks

The main aim of the present section is to enable an impression to be formed of the accuracy and width of applicability of the calculation procedure. For this purpose predictions are compared with well-documented experimental data. The successes and failures of the procedure are appraised and conclusions drawn for improving the physical or mathematical (numerical) assumptions incorporated into the procedure. In making comparisons with experiment, we must bear in mind the uncertainty which is associated with each set of measurements against which predictions are compared.

In addition to making comparisons with experiment some further calculated quantities are presented (for which no corresponding measurements are available) to provide a more complete picture of the flow field.

It was mentioned in Chapter 4 that an initial attempt to treat the near field region as parabolic was not successful since the generated solution of the flow field was not realistic. Examination of the solutions obtained showed that the failure was associated with some of the assumptions of the parabolic procedure, mainly:

(i) that the streamwise pressure gradient is known and equal to the free-stream one, and

(ii) there is no upstream travel of pressure effects.

While assumption (ii) neglects any upstream effect which the injection is having on the oncoming mainstream flow (in reality the injection would deflect the near wall mainstream fluid upwards and

and sideways), assumption (i) is totally unrealistic because by taking the streamwise pressure gradient as known (in the cases examined, zero) we do not permit the mainstream fluid above the hole to accelerate; consequently, the parabolic numerical procedure creates unrealistic high lateral pressure gradients which (for reasons of satisfying the continuity equation) create excessively high lateral velocities (much higher than the injection velocity itself). This unrealistic velocity field on the cross-stream plane creates unreasonably high lateral spread of the jet, which is not in qualitative agreement with physical reasoning nor with simple prediction models, i.e. the point-source model.

Consequently work was undertaken to apply a semi-elliptic treatment to the same flow field for which unrealistic solution was obtained with the parabolic procedure. The results obtained with the two distinct methods are selectively compared in Appendix C. From the semi-elliptic treatment solution it becomes apparent that the injection creates streamwise pressure gradients which are

- (i) by far non uniform over the cross-stream plane
- (ii) of the same order of magnitude as the lateral ones.

After the evidence that the near the injection hole flow field is indeed of the semi-elliptic type, work was undertaken to apply the developed numerical semi-elliptic procedure to various discrete-hole cooling geometries for laminar flow (laminar flow can indeed occur over the blades of modern turbines due to severe streamwise pressure gradients). The results of this investigation have been published, Bergeles et al (48) and it was demonstrated there that the procedure was predicting successfully the flow field for the test cases it was applied (a copy of ref (48) is provided as Appendix D).

The study to be presented in this chapter is concerned with a more difficult case namely that of turbulent flow.

The discrete-hole flow field of the present investigation was calculated in two steps. The region near the injecting hole was predicted treating it as a semi-elliptic region while that far from the injection hole was predicted treating it as a parabolic region (using as initial conditions the flow field calculated at the last cross-stream plane of the semi-elliptic region). The location of the change over from the semi-elliptic to parabolic region was found by making preliminary computer runs to discover how far behind the hole the streamwise pressure gradient became almost uniform over the cross-section (the pressure being less than 0.05 of that prevailing at the upstream edge of the hole plane). The conclusion from these preliminary runs was that beyond approximately five hole diameters from the upstream edge of the injection hole the pressure field became almost uniform on cross-stream planes.

The work presented in this chapter is organised in three main parts.

(i) Validation of the procedure for injecting through a single hole;

(ii) Validation of the procedure for injection through a row of holes; and finally

(iii) Validation of the procedure for injection through a double row of holes.

5.2 Preliminary calculations

5.2.1 Grid dependence tests

A preliminary investigation was carried out to explore the grid-dependence of the predicted solutions. Flow conditions corresponding to the experiment of Ramsey and Goldstein (31) for a blowing rate $M = 0.1$, $\alpha = 90^\circ$ were selected for this purpose. Figure 58a shows typical results for three different grid distributions, 10×10 , 15×15 and 19×19 grid nodes in the vertical and lateral (y, z) directions respectively; the forward step in all these cases was 0.1 hole diameters. The location of the second grid node in the y direction was fixed at the same position for all the grid distributions ($y^+ = 60$ at $y/D = 0.08$) while the grid spacing in both vertical and lateral directions was non-uniform as shown in Figure 58b. From Figure 58a it can be seen that while a 10×10 grid gave an almost grid independent solution for the streamwise velocity profiles, a 19×19 grid was required for the lateral velocities.

The effect of the forward step size on predictions emerges from Figure 58c. Predictions are shown for the three different forward step lengths employed 0.05, 0.1 and 0.2 of a hole diameter. It is seen that forward step lengths of 0.1 and 0.05 of the hole diameter give identical results, differing slightly from those obtained with a forward step length of 0.2 D.

5.2.2 Effect of the velocity boundary condition over the hole

In view of the experimental finding in the present work that the secondary injection profile is highly distorted (Section 3.1.1) numerical exploration was undertaken to examine the effect of the

secondary velocity profile in the predicted flow field. Figure 58d shows the secondary injection profiles employed in the numerical exploration and Figure 58e shows their effect on the streamwise velocity profile downstream of the injection hole along the center-line ($z/D = 0.0$). Evidently the actual secondary injection profile for this particular blowing rate and δ/D does have an influence of up to 20% on the predicted flow field immediately behind the injection hole.

5.2.3 Effect of the non-isotropic viscosity model

The final numerical exploration was associated with the turbulence model employed. In the standard form of the $k \sim \epsilon$ model there is the assumption that the turbulent viscosity is isotropic. On applying this form, the predictions showed (in comparison with experimental data) that the fluid in the centre-line region behind the hole was not accelerating fast enough and also that the fluid off the centre-line at lateral locations was insufficiently affected by injection. In the central region too high levels of effectiveness were predicted while the lateral spread was substantially too low. The above behaviour threw doubt on the adequacy of the isotropic viscosity assumption; subsequently theoretical work was undertaken to analyse the non-isotropic character of viscosity which led to the non-isotropic viscosity formula referred to in Section 4.2.

Figure 59a shows the substantial effect which the non-isotropic viscosity assumption has on the velocity profile while Figure 59b shows the effect on the film cooling effectiveness; the flow is that investigated by Ramsey and Goldstein (31).

5.3 Some computational characteristics of the semi-elliptic procedure

5.3.1 The rate of convergence

The semi-elliptic procedure is of the iterative type. Thus criteria for convergence must be set out. Various checks were made as iteration proceeded. Mainly they were the sum of the absolute values of the mass imbalances for each computational cell for the whole flow domain, the level of the pressure corrections in comparison with the pressure prevailing at the same location and the values of the velocity and temperatures field at monitoring locations. With the mass imbalance criterion the solution was regarded as converged when the imbalances had fallen to about 1% of the total flow rate.

Figure 60 shows the typical pattern of convergence of the mass sources ($\alpha = 30^\circ$, $M = 0.3$, $\delta/D = 0.122$, flow conditions those of the present study, single hole) when the initial guess of the pressure field was $p = 0$. Between 70 and 100 iterations in the axial direction were needed for full convergence. However it was discovered during the experimental part of the present study that the surface pressure fields induced by injection for various blowing rates are almost similar if non-dimensionalised with the mean product of the injection velocity and the free stream velocity. This facilitated obtaining better initial guess for the pressure field. In particular the pressure field which was obtained for a studied case was used, appropriately modified by a scaling factor, (the ratio $(v_j u_\infty)_{\text{new}} / (v_j u_\infty)_{\text{old}}$) as the initial guess for the pressure field of the new test case. This technique reduced the number of iterations required for a converged solution by a factor of 2.

On Figure 60 is also shown the rate of convergence for a test

case (similar to that mentioned earlier) but for blowing rate $M = 0.5$ in which the pressure field of the $M = 0.3$ case was used as initial condition for the higher blowing rate case $M = 0.5$. Evidently the number of iterations required for converged solution was halved, as was likewise the computing time.

5.4 Presentation of the computations - Injection through a single hole

5.4.1 Injection through a single hole 90° to the mainstream flow

In this section the predicted behaviour of the flow field created by the interaction of single jet at 90° to a mainstream is compared against experimentally documented flow fields. The experiments which form the basis of this comparison are those obtained in the present study, (hereafter referred to as the IC measurements) as well as some of those obtained by Ramsey and Goldstein (31).

(1) IC measurements: $\alpha = 90^\circ$, $M = 0.046$, $\delta/D = 0.048$, $Re_D = 3.8510^4$

Let us start with a comparison shown in Figure 61a of the predicted and measured velocity profiles along the center line downstream of the injection hole as well as along a line parallel to it at $z/D = 0.36$. The behaviour of fluid near the wall initially decelerating and subsequently accelerating is very accurately predicted, the maximum discrepancy occurring at $x/D = 5.28$. Above the wall at location $y/D > 0.2$ the predictions show that the effect of injection spreads more slowly towards the external layer of fluid than experiments suggest. However, the maximum discrepancy between prediction and measurements is less than 8%. The error analysis of the measurements shows that the accuracy of the measurements is approximately

2-5%; thus the discrepancy is not substantially larger than the experimental uncertainty.

The fact that the near-wall fluid velocity is consistently slightly underpredicted might suggest that the momentum diffusion coefficient in this region (turbulent viscosity) is slightly underestimated by the turbulence model which diminishes the rate of spread of the jet above the plate, and consequently reduces the rate at which the near wall fluid gains momentum at the expense of the external layer fluid.

Figure 61b shows comparison between the predicted values of effectiveness and the measured ones. The comparison suggests that at lateral location $z/D = 0.64$ and at $x/D = 1.4$ there is a discrepancy between prediction and measurements of about 20% while at all the other locations the predicted values are in very good agreement with the measurements. The reason for this discrepancy is not clear since the error analysis of the effectiveness measurements indicate that for a level of effectiveness of about 5% the uncertainty of the measurements is as high as 20% (comparable to the discrepancy observed above).

The success of predicting the flow field and in particular the effectiveness field is an indication that the non-isotropic $k - \epsilon$ model has acquitted itself successfully in this test case.

(2) IC measurements: $\alpha = 90^\circ$, $M = 0.1$, $\delta/D = 0.048$, $Re_D = 3.85 \cdot 10^4$

Figure 62a shows the predicted and measured values of the surface static pressure distribution. From the numerical point of view the pressure is the most sensitive of the hydrodynamic variables to the grid distribution and to the level of convergence (i.e. with the employed

criterion of convergence . the velocity field is not expected to change more than 0.5% in the last 10 iterations. At this level of convergence the pressure field can be expected to change up to 5% if the iterations were continued. If as criterion of convergence was required 1% accuracy in the pressure field, this might increase the computer time more than twice!) The level of agreement between the predicted pressure field and the measured one can be regarded as satisfactory if we recall the error associated with the pressure measurements and the fact that the contours are formed from the measurements by interpolation of the pressures taken at the pressure tappings.

Figure 62b shows streamwise velocity profiles along $z/D = 0.0$ and $z/D = 0.64$ lines. It can be seen that the predicted velocity field is in very good agreement with the experiments, the maximum discrepancy being less than 5% of the local velocity. Again it is clear that the injection does not spread out as much as the experiments suggest in accordance with the observed behaviour for lower blowing rate, in the previous case.

However the procedure is less successful in predicting the effectiveness as shown in Figure 62c. While along lateral locations $z/D = 0.36$ and $z/D = 0.64$ the predicted effectiveness coincides with the measured ones, near the centre line ($z/D = 0.0$ and $z/D = 0.18$) and immediately behind the hole the effectiveness is over predicted. Indeed, in contrast to the measurements the predictions fail to display any region where $\eta_{z/D = 0.18} > \eta_{z/D = 0.36}$.

The reason for this lies in the treatment of the flow as predominantly one way; this does not allow the presence of any reverse

flow in the streamwise direction. The measurements have shown (Chapter 3) that behind the injection hole there is reverse flow. The calculation procedure also indicates the presence of reverse flow behind the hole but since this feature is not allowed by the procedure, (as explained in Section 4.1.1) this back flow region is patched over by effectively substituting an almost stagnant region. In reality, the reverse flow region brings low effectiveness/^{fluid}from downstream towards the downstream edge of the injection hole; this reduces the level of effectiveness behind the hole. This mechanism is suppressed by the semi-elliptic procedure which explains why the effectiveness behind the injection hole is overpredicted.

Further downstream the agreement between predictions and measurements is satisfactory, suggesting that the reverse flow has only a localised effect on the flow field.

The existence of reverse flow for $M = 0.1$ and $\alpha = 90^\circ$ poses a serious limitation to the width of applicability of the developed numerical procedure, since for blowing rates greater than 0.1 a fully elliptic treatment is required for the region behind the injection hole. This matter will be discussed further in the final chapter of the thesis.

- (3) Measurements by Ramsey and Goldstein (31): $\alpha = 90^\circ$, $M = 0.1$,
 $\delta/D = 0.03$, $Re_D = 0.87 \cdot 10^5$

While the cases presented previously were a test of the procedure in the immediate vicinity of the injection hole, the data by Ramsey and Goldstein (31) extend far downstream from the injection hole; these measurements therefore provide a test of both the semi-elliptic and parabolic procedures. Computations with the semi-elliptic procedure

extended up to $x/D = 5$ while the region beyond this was treated with the parabolic scheme.

Figure 63 (a-b) shows detailed effectiveness measurements by Ramsey and Goldstein (31) and predictions of the present procedure for several locations in the flow field. It is evident that on the centre-line region (for z/D up to 0.375) the level of the wall effectiveness is very accurately predicted, as is the vertical spread of the jet. For $z/D > 0.5$ it is seen that while the vertical spread of the jet is again accurately predicted the wall effectiveness is underpredicted. One basic fact which gave us doubts about the accuracy of the measured wall effectiveness in this region is the steep wall temperature gradient adjacent to what was nominally an adiabatic plate; one would expect to find a zero wall temperature gradient. A second fact which suggests indirectly that the test plate of (31) was not perfectly adiabatic (and hence that conduction errors were present in the measurements), is that, in comparison with the present mass transfer data (Section 3.1) the reported film cooling effectiveness of (31) is much higher. Consequently no assessment of the predictions can be made in this region. It should also be noticed that this difference between predictions and measurements near the wall is a maximum near the injection hole and reduces gradually downstream.

Figure 63c compares the wall film cooling effectiveness predictions with the measurements for the far downstream of the injection hole flow field which was treated as parabolic. The agreement is very satisfactory; it may be recalled from this figure as well as from Figure 59b, that the satisfactory agreement achieved is due in large measure to the use of a non-isotropic viscosity.

Some further characteristics of the flow field

We now examine some predicted characteristics of the flow field for which there are no experimental data available and which help to explain features of the flow already presented.

The existence of a secondary vortex behind the injection hole has been detected experimentally for high blowing rates but not for low values because the secondary motions are very weak. Let us now examine the flow field on planes cross-stream to the flow direction with the downstream distance, to see if these vortices exist.

Figure 64 (a-b) shows the flow patterns upstream of and over the hole for 90° injection and $M = 0.1$. It is evident that due to the injection, the upstream fluid at $x/D = -0.2$ is pushed upwards and, principally, sideways: this is clear evidence of the semi-elliptic character. Further downstream at $x/D = 0.4$ (which is located over the injection hole) can be seen the movement of the injected fluid as it spreads upwards and sideways, the maximum velocity being equal to the injection one. At $x/D = 1.4$ the beginning of the formation of a vortex is evident, signalled by the inward movement of the near wall fluid*. By $x/D = 2.0$ the vortex has been completely formed, however the sideways moving fluid has greater velocity than the fluid within the vortex. Further downstream, $x/D = 2.65$ and $x/D = 3.40$, the velocity of the sideways moving fluid is decreased while the dimensions of the vortex increase as well as the fluid velocity within it. At locations further downstream the vortex decreases in size and the fluid velocity becomes weaker and weaker.

* The symbol z over the velocity vector on the figures indicates that the length of the vector is out of scale.

At a location $x/D = 2.65$ and $z/D = 0.4$ the inward moving near-wall fluid has a lateral velocity of $0.04u_{\infty}$ and a streamwise one of about $0.3u_{\infty}$. Consequently the low effectiveness fluid located at $x/D = 2.65$ and $z/D = 0.4$ will be brought to the centre-line region by the action of vortex where it will contribute to the dilution of the jet over a streamwise distance of, $\frac{0.4D}{0.04u_{\infty}} \cdot 0.3u_{\infty}$, i.e. approximately equal to 3 hole diameters; it may thus be concluded that immediately downstream of the hole the high rate of dilution of the centre-line region of the jet is due to the mixing of the jet with mainstream fluid which has passed over the injection hole (and also to the backward moving fluid in the region of recirculation). Further downstream the vortex contributes to the jet dilution. It is expected that, as the blowing rate increases, the centre-line streamwise velocity behind the hole will decrease, the strength of the vortex will increase (due to higher vorticity added into the flow from injection) and hence the dilution of the jet due to the vortex action will start much earlier and will be stronger.

Now let us examine some turbulence characteristics of the flow field. Figure 65a shows the turbulence intensity distribution on cross-stream planes at various downstream distances. The dotted lines on the figures represent the locations of 0.05 and 0.005 turbulence intensity for the undisturbed mainstream fluid. As seen from the figures, the injection generates turbulence and at location $x/D = 0.4$ the maximum turbulence intensity, which is located at the edge of the jet emerging out of the hole, has increased to 20%. The turbulence intensity reaches a maximum level of 30% at $x/D = 1.4$ and then decreases further downstream, the high level intensity contours

shrinking and moving towards the centre-line region of the jet while the 0.05 intensity contour spreads upwards and sideways as the injection spreads over a broader region. Figure 65b shows more clearly the effect of blowing on the turbulence intensities on the symmetry plane; the region behind the injection hole being a region of very high levels of turbulence.

5.4.2 Injection through a single hole at 30° *

(1) IC measurements: $\alpha = 30^\circ$, $M = 0.1, 0.2, 0.3, 0.5$ and 1.0

Figures 66 (a-e) show the predicted streamwise velocity profiles compared with the measured ones for various blowing rates and at various locations in the flow field. For the two lower blowing rates $M = 0.1$ and $M = 0.2$, the agreement between predictions and measurements is very satisfactory at all downstream and lateral locations. As the blowing rate is increased however to $M = 0.3$ and $M = 0.5$ the agreement is less satisfactory, the maximum discrepancies occurring at location $y/D = 0.4$ above the wall.

It can be seen in the figures that in the region $y/D = 0.4$ the fluid velocity for $z/D < 0.53$ is consistently underpredicted for all the blowing rates and overpredicted at lateral locations for which $z/D > 0.53$. This behaviour suggests that the calculated lateral transport of streamwise momentum is too small with the result that the rate of transport of streamwise momentum from the faster moving fluid (at large z/D) to the center-line region fluid behind the hole is likewise too low.

* For 30° injection the hole extends from $x/D = 0.0$ to $x/D = 2.0$

The success of any numerical procedure for predicting turbulent flows lies mainly on how well the physical process of turbulence is represented by the turbulence modelling. It may be recalled that in the present study non-isotropic $k - \epsilon$ model is applied for the first time and the equation expressing this non-isotropy has been derived in Section 4.1.2 on the following assumptions.

- (i) There is local equilibrium (production of k equals ϵ)
- (ii) The only significant velocity gradients occurring in the flow are $\frac{\partial u}{\partial y}$ and $\frac{\partial u}{\partial z}$

For blowing rates $M > 0.3$ both experiments and predictions suggest that the jet emerging from the hole penetrates into the mainstream. Consequently it is expected, as the blowing rates increase, the correctness of the above assumptions will progressively deteriorate. Consequently for more accurate prediction of the flow field a more general non-isotropic $k - \epsilon$ model should be used, by employing approximate forms of the normal stresses and calculating the ratio τ_3 / τ_2 instead of adopting the simplified form (4.36). Throughout the present study however the same non-isotropic $k - \epsilon$ turbulence model was employed. The reason for doing so was to obtain a thorough impression of the range over which the formula (4.36) was applicable; and as shown by the predictions even for blowing rate $M = 1.0$ the simple relation is performing fairly satisfactorily, the maximum discrepancy being around 15% in the region where $y/D = 0.5$.

Figure 67 shows the predicted values of the wall film cooling effectiveness compared with the measurements for blowing rates $M = 0.1, 0.2, 0.3, 0.5$ and 1.0 . For the two lower blowing rates, $M = 0.1$ and $M = 0.2$, the agreement can be regarded as satisfactory except at the

lateral location $z/D = 0.80$. For levels of effectiveness of around 10% the experimental uncertainty of the measurement is quite high (around 15%); the observed maximum difference between measurement and prediction is about 20%, i.e. a roughly comparable level. When the blowing rate is increased to $M = 0.3$ (as well as for $M = 0.5$ and 1.0) the effectiveness in the center-line region just behind the hole is overpredicted. Behind the injection hole for blowing rates $M > 0.3$ experiments (Chapter 3) and the present predictions suggest that the jet lifts off the surface and mainstream fluid penetrates beneath it. Just behind the injection hole there is a very small region of recirculation, which becomes stronger for higher blowing rates. This region is very small in length and height (length = $0.2 D$, height = $0.02 D$): its existence is predicted by the procedure but as was explained earlier its effects are not accurately simulated. The height of this region corresponds to a value of $y^+ = 10$; consequently apart from the fact that the procedure employed in the study does not rigorously compute the reverse flow region, the version of the $(k - \epsilon)$ turbulence model used does not allow the near-wall node to be located closer to the wall than $y^+ = 35$ (so the grid node is in the fully turbulent region) and hence by locating the near-wall grid node at $y^+ = 35$ (y/D = 0.06) the numerical procedure can not resolve the region of recirculation which reduces substantially the level of effectiveness just behind the injection hole. (Up to $M = 0.5$ the effect of reverse flow is small and agreement is reasonable.) For a blowing rate $M = 0.5$ it is seen that the effectiveness at $z/D = 0.53$ is underpredicted by 10%. This is consistent with the lateral transport of momentum being

slightly underestimated by the solution procedure.

Figure 67b shows the predicted wall effectiveness for $M = 1.0$. It is seen that the neglect by the procedure of the reverse flow region behind the hole results in a substantial overprediction of the wall effectiveness in this region. Laterally (where there is no reverse flow) the agreement between predicted and measured values is very satisfactory. Also on the figure is shown computations, for the purpose of demonstration only, where the distance of the near wall grid point was small ($y/D = 0.03$, $y^+ = 10$); in this case the procedure did predict reverse flow behind the hole which however was patched over as described in the Appendix B. Just behind the injection hole the level of the predicted wall effectiveness has dropped substantially, in qualitative agreement with the measurements. Flow field predictions were made further downstream for $M = 1.0$ using the parabolic procedure and setting $y^+ = 35$. The predicted values of the wall effectiveness at $z/D = 0.45$ and $z/D = 0.90$ are in good agreement with the measured one except at the center-line region where the effectiveness is underpredicted, the maximum discrepancy being about 10% (this is a further consequence of the bad predictions in the semi-elliptic region).

Some further characteristics of the flow field for blowing rate $M = 0.5$

Figure 68 shows the predicted pressure field on various cross-stream planes. At location $x/D = -0.2$ the injection creates a pressure build-up upstream of the injection hole, the maximum pressure coefficient $C_{p_{max}}$ being located on the centre line and equal to 0.72. Over the injection hole at $x/D = 0.40$ the pressure close to the hole continues to increase, with $C_{p_{max}}$ reaching the value 1.0 while at the near-wall lateral locations the pressure drops as the fluid, having been

initially retarded, starts to accelerate. At a location $x/D = 0.90$ the pressure contours change drastically, the maximum pressure coefficient being only 0.26 and located above the wall. This is the result of the considerable acceleration which the near-wall fluid is experiencing due to the streamwise momentum which the injected fluid possesses. At locations $x/D = 1.4$ and $x/D = 2.0$ further downstream the negative pressure field spreads outwards as the mainstream fluid is accelerating, the minimum pressure coefficient taking the value of -1.30. Further downstream $x/D > 2.0$ the pressure recovers slowly as the flow reverts to a boundary layer type. At location $x/D = 4.10$ the pressure coefficient has minimum value $C_p = -.27$ while the $C_p = -0.20$ pressure contour has taken up a kidney shape.

Figure 69 shows the vector field on cross-stream planes at several locations through and downstream at the hole. At location $x/D = 0.90$ the injected fluid is seen to spread upwards and sideways into the mainstream. At $x/D = 2.0$, just at the end of the hole, it is seen that whilst the fluid near the centre line has significant upward movement, a vortex pair of small dimensions has already been created behind the injection hole. Further downstream, $x/D = 3.4$, the size of the vortex has significantly increased as has also the velocity associated with it, while the vortex itself is moving upwards. At $x/D = 5.9$ the dimensions of the vortex have increased whilst the velocity has decreased. Apparently the presence of the vortex pair contributes significantly to the faster dilution of the jet especially in the near-wall region since its movement brings into the centre line region of the jet fluid from the top and sides of low effectiveness.

Figure 70 shows the effectiveness contours on planes normal to main flow direction. As the injectant enters into the mainstream it spreads laterally as well as vertically. Moving downstream, at $x/D = 2.0$, the level of effectiveness increases due to the continuous injection of coolant, while the jet spreads outwards. However, the rate of spread in the vertical direction is faster than in the lateral one while a new feature appears: the inward shrink of the $\eta = 0.75$ and 0.50 effectiveness contours near the wall and at $z/D = 0.5$. This characteristic which is more pronounced at $x/D = 3.40$ is due to the presence of the vortex pair discussed earlier. At location $x/D = 3.40$ the maximum effectiveness occurs above the wall, an indication that the fluid carried by the vortex has penetrated beneath the jet as the jet has lifted off the surface. At location $x/D = 4.10$ the trend is the same with the vortex contributing to further dilution of the jet.

On Figure 70 some effectiveness contours for blowing rate $M = 0.3$ have also been plotted in broken lines. It is seen that for the latter case the maximum effectiveness lies almost on the surface, indicating that due to the lower blowing rate (lower vertical momentum) the jet is more quickly bent by the mainstream and remains in touch with the wall. At $x/D = 4.10$ in particular it can be seen that the vertical spread of the jet is considerably smaller than that for $M = 0.5$, with a slightly increased lateral spread.

Finally, we make comparison of the effect which the various blowing rates have on the hydrodynamic and effectiveness fields. Figures 71 (a-b) show the $\eta = 50\%$ and $\eta = 10\%$ contours on the $z/D = 0.0$ plane. It is apparent that the higher the blowing rate the more the jet tends to penetrate into the mainstream. Figure 71c shows the

contour of maximum effectiveness which can be associated with the jet trajectory. Evidently for blowing rates $M = 0.1$ and 0.2 the jet is bent staying in touch with the surface while for $M = 0.3$ it lifts off though reattaching quickly to the surface. For a blowing rate of $M = 0.5$ the jet penetrates into the mainstream, still residing within the boundary layer while for $M = 1.0$ it has penetrated well into the mainstream flow.

Figures 72 (a-b) show the wall film cooling effectiveness. Evidently blowing rates $M = 0.2$ and $M = 0.3$ give almost the same lateral spread; however $M = 0.3$ results in a higher level of effectiveness in the centre-line region, while for a blowing rate of $M = 0.5$ the coverage of the surface is not as complete as with $M = 0.3$.

Figure 73 shows comparison of the momentum losses (expressed by the change of momentum thickness) associated with fluid near the centre line of the jet. Normal injection $M = 0.1$ blowing rate produces the highest momentum losses for all the blowing rates (of the inclined injection) while a blowing rate of $M = 0.5$ in fact adds momentum into the mainstream flow.

The fact that the momentum losses are high for normal injection is very important since aerodynamic losses decrease the attainable efficiency of the turbine.

5.5 Injection through a single row of holes

The numerical procedure is here validated for the single row arrangement by comparing the predicted flow field against experimentally documented flows of Eriksen (29), Pedersen (18) and those from the present study.

5.5.1 Measurements by Eriksen (29) $\alpha=35^\circ$, $M=0.2$, $s/D=30$, $\delta l/D=0.149$

$$\underline{R_{eD} = 0.4410^5}$$

All the measurements taken by Eriksen (29) refer to adiabatic wall film cooling effectiveness for the region far downstream of the injection hole ($x/D \gg 5.0$).

Figure 74 shows the predicted values of the wall film cooling effectiveness for the parabolic region compared with measurements. It is seen from the figure that for $x/D \gg 15.0$, the predicted values agree very well with the measured ones. At downstream locations $x/D \leq 15$ the effectiveness along the centre line, $z/D=0.0$, is underpredicted by about 5%. The experimental uncertainty of the measurements given by Eriksen is around 2% and the centre line effectiveness measurements have been corrected for errors due to wall conduction. At lateral location however ($z/D \gg 1.0$) the effectiveness is underpredicted by approximately 20%. Because of the low level of effectiveness at these lateral locations however the accuracy of these measurements is questionable, certainly it is not $\pm 2\%$. Moreover, the existence of heat conduction within the test plate may have contributed to higher values of effectiveness at $z/D \gg 1.0$ and lower values at $z/D=0.0$ (which are corrected by Eriksen). The tendency to underpredict the effectiveness at lateral location was also found for the similar experiments for a single hole $\alpha=90^\circ$

by Ramsey and Goldstein (31), section 5.4.1. Further downstream where we may expect conduction in the test plate not to affect significantly the measurements, the agreement is satisfactory.

5.5.2 Measurements by Eriksen (29), $\alpha=35^\circ$, $M=0.5$, $s/D=3$, $\delta l/D=0.149$, $R_{eD}=0.44 \cdot 10^5$

Figure 75 provides a comparison between the predicted and the measured wall film cooling effectiveness for a higher flowing rate of 0.5. For the flow field far downstream, $x/D \gg 20.0$, the agreement between calculations and experiments is satisfactory, however for $x/D < 20.0$, particularly for $z/D \gg 1.0$, the predicted values of wall effectiveness lie below the measurements. It must be noted that the same behaviour was found for a blowing rate of $M=0.2$ and the discrepancy is probably due to heat conduction in the test plate of Eriksen's experiment.

A characteristic of the measurements which is confirmed by the predictions is the fact that beyond $x/D=40.0$ the effectiveness at lateral locations start to become greater than the values on the centre line. The reason for this behaviour will emerge when the flow field on cross stream planes is presented later.

5.5.3 Measurements of Pedersen (18), for non-unity density ratio $\alpha=35^\circ$, $M=0.5$, $\rho_c/\rho_\infty=2.0$

Pedersen (18) in his study used mass transfer analogy to measure the film-cooling effectiveness. Obviously his measurements are free from heat conduction errors. The chosen test case for the validation of the numerical procedure has a density ratio of $\rho_c/\rho_\infty=2.0$, which is close to the value which would be found in practical situations.

As in the Eriksen's experiment, Pedersen's measurements are for the far field and the computation again proceeds in two steps.

Figure 76a shows a comparison between predicted centre-line and laterally averaged wall effectiveness with measured ones. It is seen that the agreement is very satisfactory throughout the flow field. Figure 76b shows a comparison for the lateral variation of wall effectiveness for various downstream locations. It may be noted that at $x/D=12.0$ and at off-centre locations the agreement between prediction and measurements for Eriksen's flow was unsatisfactory in contrast with the present case.

5.5.4 IC measurements, $\alpha=30^\circ$, $M=0.25$, $s/D=2.667$, $\delta l/D=0.095$

Figure 77 shows predicted values of wall film cooling effectiveness compared with measured ones at various lateral and downstream locations. The procedure predicts accurately the wall effectiveness for the far downstream flow field. In the immediate vicinity of the injection hole however the effectiveness at $z/D=0.53$ is slightly underpredicted, whilst at $z/D=0.8$ it is underpredicted by approximately 10%; this is comparable with the experimental uncertainty in this region. The fact that the effectiveness near the centre line is slightly overpredicted while at $z/D=0.53$ it is slightly underpredicted might suggest that the exchange diffusion coefficients (viscosity or mass diffusivity) are slightly undercalculated.

A second possible explanation for these small discrepancies might be the uncertainty in the secondary injection velocity profile which becomes greater as the boundary layer thickness decreases (a topic to be taken up in the next paragraph).

5.5.5 IC measurements, $\alpha=30^\circ$, $M=0.5$, $s/D=2.667$, $\delta l/D=0.095$

Figure 78a provides a comparison between predicted values of wall effectiveness and measurements for the semi-elliptic region of the flow field. The injection velocity over the hole is prescribed as uniform in the predictions as was the practice for all the single-row calculations. From the figure it is evident that the effectiveness is satisfactorily predicted at lateral locations $z/D=0.8$ and $z/D=1.07$ but that it is highly underpredicted in the region $z/D \leq 0.53$.

Comparison of the present predictions ($\delta l/D=0.095$) with those obtained for injection through a single hole with thicker boundary layer ($\delta l/D=0.122$) reveals but little changes in the effectiveness, in contrast to the experiments which indicate substantial differences (Section 3.2). This means that the effect of the thinner boundary is not properly accounted for by the numerical procedure. It may be recalled from the predictions for a single 90° hole (Section 5.1) that numerical exploration showed how the actual shape of the velocity profile of the jet influenced the flow field downstream of the injection hole.

However, for the flowing rate examined there, $M=0.1$, the influence was not particularly strong for that boundary layer thickness. A similar numerical exploration was undertaken in the present study to study the effect of the injection velocity profile on the flow field, in an effort to trace the reason for the high discrepancies between measurements and predictions.

On figure 78a there is also plotted the prediction obtained assuming that over the injection hole the velocity profile of the jet is a trapezoid, shown in figure 78c (with the jet still emerging

from the hole at 30°). It can be seen that while the effectiveness is not influenced well off centre line, there is substantial effect along the centre line itself. Next the assumption that the jet emerging from the injection hole at 30° was questioned; figure 78b containing the same data, shows results obtained with two different injection profiles defined in figure 78c.

Evidently the injection velocity profile for $\alpha_1 = 20^\circ$, velocity profile 3, results in a much higher level of centre line region effectiveness, in accord with experiments, with some reduction of the lateral spread of the jet; whilst injection profile 4 gives almost the correct levels of lateral effectiveness but still predicts higher centre line effectiveness than profiles 1 and 2, figure 78c.

In conclusion, from this numerical exploration, it becomes evident that the velocity profile of the jet does have significant effect on the flow field immediately after the hole, the effect decreasing with the downstream distance. This suggests that the effect of boundary layer thickness on the flow field comes mainly through altering the velocity profile of the emerging jet.

Figure 78d shows the predicted values of the effectiveness for the parabolic region of the far field, where the initial conditions used were those obtained with the velocity profile 4, figure 78c. The agreement can be regarded as still satisfactory except at the beginning of the computations where uncertainties in the initial conditions have a significant effect.

5.5.6 Some further characteristics of the flow field

To provide a better understanding of the flow field we shall

present below some further predicted characteristics.

Experiments reported in chapter 3, revealed that while the velocity profile of the secondary jet is highly distorted, on examination it appears that the total pressure of the secondary jet over the hole is almost constant. In the present study one of the various boundary conditions applied over the injection hole was that P_{tot} is constant,

$$P_{tot} = P_s^* + \frac{\rho}{2} (MU_\infty)^2$$

The above relation is used to calculate the resultant injection velocity over the hole $V_j = \sqrt{\{(P_{tot} - P_{s,hole}) / (\rho/2)\}}$ and hence the u_e and v_e distribution.

Applying this boundary condition, the velocity of the jet is no-longer constant but highly distorted. Figure 79 shows the resultant profile for the IC measurements at $M=0.25$. It is seen that most of the jet emerges from the downstream half of the injection hole, whilst the minimum of velocity occurs along the centre line of the hole. This behaviour is in qualitative agreement with the measurements of the present study reported in chapter 3 as well with similar measurements by Gosman and Simitovic (73) for a jet emerging from a square hole into a cross flow.

Figure 80 shows predicted streamwise velocity contours on cross-stream planes at various downstream locations for the Eriksen flow, $M=0.5$. At $x/D=-0.2$, upstream of the injection hole, the main-stream fluid is slightly decelerated; over the injection hole and near the hole, the fluid is further decelerated whilst fluid at a greater

* P_s is such that the integral of the resultant injection velocity times area over the hole gives the correct amount of coolant injected into the flow.

height above the hole is accelerated as can be seen by the position of the $u/U_\infty = 1$ contour, the maximum velocity contour being 1.04. For $z/D > 0.5$ the fluid is strongly accelerated. Behind the hole, at $x/D = 3.40$, the centre line region fluid has been strongly decelerated, the velocity minimum being located off the surface. This behaviour, which was also brought out by the experimental study, is associated with the vortex pair existing behind the hole as discussed below. Further downstream the fluid near the centre line starts accelerating at the expense of the momentum of adjacent layers of fluid and then reverts for $x/D > 25$, to a boundary layer velocity profile.

The development of the vortex pair is shown in figure 81. Upstream of injection hole at $x/D = -0.2$, the mainstream fluid is deflected upwards and sideways. The figure should be contrasted with the corresponding figure 64 for the single-hole case. The presence of the adjacent injection holes here causes the sideways movement of the fluid to be transformed to an upward motion. Over the injection hole, the emerging jet spreads into the mainstream fluid, while at $x/D = 3.4$ and $x/D = 5.2$ the familiar vortex has already been formed behind the injection hole. This vortex brings high momentum fluid from the region between the holes to the centre line region thereby causing the near wall fluid there to acquire higher momentum than the fluid above it. As a result, the minimum velocity behind the hole exists away from the wall, as shown in figure. However, this "pumping" action of the vortex also brings to the centre line region fluid of low effectiveness with the result to contribute to faster dilution of the emerging jet. The circulatory motion of the vortex persists further downstream up to $x/D = 40$, where the motion starts to

become disordered commencing from the centre line region. At location $x/D=53$ is seen that while an inward movement of the fluid from the sides still exists, the movement is now such that the fluid from the top layers of the centre line region is brought downwards to the wall. This will have as a result, since the near wall fluid has achieved the same effectiveness by this location, figure 75, to decrease the centre line wall effectiveness to values lower than the adjacent ones, i.e.

$$\eta_{z/D=0} < \eta_{z/D>0}$$

Finally, figure 82 shows effectiveness contours on cross-stream planes at various downstream locations. At $x/D=0.90$ the effectiveness contours spread roughly evenly in the y and z directions, while at $x/D=3.4$ and $x/D=5.2$ the upward spread of the contours continues, but the lateral spread, which from the practical point of view is the most important (since it determines the lateral extent of coverage has considerably decreased due to the action of the vortex pair. At $x/D=5.2$ it is clearly seen that the effectiveness maximum lies off the wall. These are indicating that the fact that the jet lifts off the surface. Also shown are some effectiveness contours for the same blowing rate, but for Pedersen's flow where the density ratio is 2. The denser jet remains closer to the surface, the upwards spread of the effectiveness contour being reduced, while its lateral spread is increased due to the weaker vortex pair behind the injection hole.

5.6 Injection through a double row of holes

5.6.1 Introductory remarks

From the presentation of the results of the validation studies for the single hole and single row of holes DHC arrangements

it may be concluded that the numerical procedure here developed can be safely used as design tool for such cooling geometry. In practice however in the near future it may not be possible to meet the cooling requirements of the blade with these DHC arrangements; a double row of holes arrangement might be necessary or might, in any case, be judged desirable.

If the streamwise spacing of the two row of holes is large enough, i.e. $S_x/D=10.0$, then the jets from the second row of holes would have scarcely any effect on the jets from the first row; consequently the procedure, could safely be used in calculating the flow in stepwise manner, by first computing the flow field from the first row of holes and then carrying the solution further downstream to cover the second row.

On the other hand, for small spacing between the two rows, the injection from the second row affects the flow field of the first and consequently the calculation of the flow field should proceed over a region which would include both rows, treating the entire region as semi-elliptic. At sufficiently large distances downstream from the second row of holes the flow will of course become parabolic.

In the present study the procedure has been tested for the more complex flows, where the spacing between the two rows is small, using for the validation the experimentally documented flow fields of Jabbari (25) and the present study.

5.6.2 Measurements by Jabbari (25), $M=0.334$, $s/D=3.0$, $S_x/D=2.6$

Figure 83a shows the distribution of film cooling effectiveness in the semi-elliptic region of the double row. The effectiveness

along the two centre lines ($z/D=0.0$ and $z/D=1.50$) reduces uniformly with downstream distance; the same behaviour takes place along the line $z/D=1.0$ up to distance $x/D=8.0$ after which the effectiveness slightly increases due to the mixing of the jets from the two rows. An important feature of figure 83a is the effectiveness distribution along $z/D=0.50$, for the rate of decrease of the effectiveness here is substantially higher than that which would prevail along the same line if the second row of holes was absent. The effectiveness along $z/D=0.50$ reaches a minimum around $x/D=3.8$, which is almost midway to second row of holes, and then recovers very quickly downstream. The explanation for this behaviour will be given later: it is the result of an influence of the downstream row of holes on the upstream one.

Jabbari's (25) measurements refer to the far-downstream flow field which was here calculated using the parabolic procedure. Figure 83b presents a comparison of the predicted values of effectiveness with the measured ones. Along the line $z/D=1.50$ the predicted and the measured values agree satisfactorily; along the other lines slight differences occur, the maximum overall discrepancy being about 8%, which is comparable with the experimental uncertainty of the measurements. A point to notice is that for the region $x/D < 14$ the predicted value of effectiveness along $z/D=0.50$ is lower than along $z/D=0.0$, in contrast to the measurements, suggesting that the predicted recovery of effectiveness along $z/D=0.50$ should be slightly faster. It is seen that near two dimensionality occurs by $x/D=30$.

5.6.3 Measurements by Jabbari (25), $M=0.45$, $s/D=3.0$, $S_x/D=2.6$

When the flowing rate is increased from $M=0.334$ to $M=0.47$ the general behaviour of the flow field is similar to that described

above. In particular, figure 84a shows the predicted wall film cooling effectiveness for the semi-elliptic region. The level of effectiveness along $z/D=0$, 1.0 and 1.50 has increased in comparison to $M=0.334$ except along the $z/D=0.50$ line, where the minimum effectiveness, occurring at $x/D \approx 3.5$, is lower than the corresponding one for $M=0.334$, this indicating that the upstream effect of the second row of holes is stronger.

Jabbari's measurements of wall effectiveness again refer to the region $x/D > 10$ which was computed using the parabolic procedure. The predicted effectiveness is compared with the measurements on figure 84b. The level of agreement is generally very satisfactory, small discrepancies which are comparable with the experimental uncertainty of the measurement occurring as for the lower blowing rate, at $x/D \approx 1.0$.

Finally, figure 85 shows predicted and measured laterally averaged wall film cooling effectiveness for the two blowing rates, $M=0.334$ and $M=0.47$; the agreement is good, slight discrepancies occurring in the region $x/D \approx 10$.

From these calculations it is concluded that the procedure predicts correctly the wall film cooling effectiveness in the region far from the holes; however up to now there is no direct evidence about its performance in the immediate vicinity of the holes. Below some of the measurements taken in the present study, and reported in chapter 3, will be used for the validation of the (semi-elliptic) procedure in this region.

5.6.4 IC measurements, $\alpha=30^\circ$, $M=0.25$, $s/D=2.667$, $S_x/D=3.5$

Figure 86a shows the predicted wall film cooling effectiveness along the lines $z/D=0.0$, 0.53 and $z/D=1.33$. It is seen that the effectiveness along $z/D=0.0$ and 1.33 is satisfactorily predicted

but that along $z/D=0.53$ recovers much more quickly than the experiments suggest. As will be seen later, in this region, $x/D=4.50$ and $z/D=0.53$, the lateral velocities (on the cross-stream plane) are almost zero, hence the lateral spread of the jet is predominantly controlled by diffusion. The fast recovery of the effectiveness along $z/D=0.53$ line suggests that the effective diffusion coefficients in the z direction are overestimated. The conclusion is that whilst for the single row of holes geometry it was shown that the diffusion coefficients in both y and z directions were accurately calculated, in the present case, due to the stronger interaction between the jets (mainly from the first and second rows), the level of non-isotropy of viscosity in the y and z directions is reduced, hence resulting in smaller lateral diffusion.

Also shown in figure 86a are predictions where an isotropic viscosity was assumed. It is seen that when this is done recovery of the effectiveness along $z/D=0.53$ is substantially reduced, which accords with the experiments. However, the level of effectiveness along the $z/D=0.0$ and $z/D=1.33$ locations, is overpredicted. The above characteristics suggest that the viscosity along the $z/D=0.0$ and $z/D=1.33$ remains non-isotropic, as for the single row geometry, while as the region $x/D=4.50$ and $z/D=0.53$ is approached the flow develops a rather less non-isotropic character.

Figure 86b shows the predictions obtained with the "standard" non-isotropic version of the (K- ϵ) model for the semi-elliptic and parabolic far downstream regions of the flow field.

5.6.5 Some further characteristics of the flow field

Below are presented some further predicted characteristics of the flow for the conditions $M=0.47$ of the Jabbari's experiments.

Figure 87 shows contours of streamwise velocity (expressed as u/u_∞) on cross-stream planes at various downstream distances. At $x/D=-0.20$, just upstream of the first row of holes, the mainstream fluid is decelerated and the region of deceleration spreads laterally as the flow proceeds over the injection hole to $x/D=0.90$. However, at this location the off centre line fluid ($z/D > 0.5$) is strongly accelerated as can be seen from the 0.80 and 1.0 velocity contours. At $x/D=2.40$ (a plane downstream of the first row of holes but just $-0.2D$ upstream of the leading edge of the second row) the near-wall fluid along $z/D=0$ is further decelerated because of the adverse pressure gradient behind the hole. The same behaviour occurs along $z/D=1.50$ because of the injection from the second row of holes. However, the fluid in the region $0.5 < z/D < 1.0$ is substantially accelerated as can be more clearly seen at $x/D=3.40$ (plane over the second row of holes); this is simply a consequence of mass conservation. At the latter plane the fluid along $z/D=0.0$ starts to accelerate (the 0.3 velocity contour having disappeared) while the fluid along $z/D=1.50$ keeps decelerating due to the injection from the second row of holes.

At the plane $x/D=4.90$ the behaviour is very similar to that for $x/D=3.40$ with the difference that region of the fastest moving fluid at $z/D=0.8$ has shifted towards $z/D=0.50$ region, indicating that the fluid is deflected sideways by the pressure field provoked by the downstream injection. Further downstream beyond the second row at $x/D=8.40$ the fluid along $z/D=1.50$ line, accelerates at the expense of the momentum of the fast moving fluid near $z/D=0.5$, which is gradually

decelerating.

The interaction between the two rows of holes already evidenced can be more clearly perceived by figure 88, which shows the vector fields on cross-stream planes at various downstream distances.

Figure 88a shows the velocities in the plane of the cross-section at $x/D = -0.2$ (just upstream of the first row of holes). It is clear that the now-familiar sideways deflection of the mainstream fluid occurs due to the upstream effects of injection, the maximum lateral velocity created being around 0.04 of the free stream velocity. At $x/D = 0.9$, figure 88b, the injected fluid from the first row of holes spreads upwards and sideways into the mainstream fluid (note the similarity with figure 81 for injection through a single row). At $x/D = 2.40$ (0.60 hole diameters after the first row of holes and -0.2 hole diameters upstream of the second row) the familiar vortex has been formed as in the single hole and single row situations, but there is here an additional characteristic at the $z/D = 1.50$ location; that of the sideways deflection of the mainstream fluid due to the upstream influence of the second row of holes. Comparison of the magnitude of the maximum near wall velocity in the vortex with the minimum velocity for a single row (shown in figure 81) reveals that the former is twice as high as the latter because of the abovementioned effect of the second row. However, the width of the vortex is smaller in the present case, because the downwash movement of the vortex has been reduced by the opposing upwards movement of the deflected fluid in front of the downstream holes.

At the plane $x/D = 3.40$, through the downstream holes, it can be seen that, due to the injection of fluid with vorticity of an opposite sign the vortex attached to the first row of holes has almost been

destroyed. Another noteworthy feature of figure is the strong lateral movement of the near-wall fluid, which is mainly of low effectiveness and will therefore promote the dilution of the jets from the upstream row. Further downstream beyond the second row the following features are noteworthy :

(i) A second vortex has been formed, attached to the second row of holes

(ii) The vortex attached to the first row of holes which was almost erased at $x/D=3.40$ persists, but is much weaker than the second vortex.

Further downstream, for $x/D > 5.9$ the strength of the second-row vortex is reduced to a level nearer to that of the first-row one, whose strength is also diminishing. The rate at which the vortex motion weakens is faster than in the single row case due to the opposite sense of rotation of these two vortices.

Finally, figure 89 shows the effectiveness contours on cross-stream planes. At plane $x/D=0.90$ the jet spreads evenly laterally and upward in the same way as for the case of single row, figure 82, while at $x/D=2.40$ the level of wall effectiveness is much smaller than the levels, at corresponding locations, for a single row. This behaviour is due to the sideways deflection of the low effectiveness mainstream fluid by the downstream row of jets.

At location $x/D=3.40$ it is seen that maximum effectiveness occurs above the surface, indicating that the jet from the first row of holes has clearly penetrated into the mainstream; the jet emerging from the second row of holes spreads outwards and the fluid in the region $z/D=0.8$ (which was shown in figure 87 to have the maximum

acceleration) has the smallest effectiveness. Further downstream, beyond the holes at $x/D=5.40$, the effectiveness contours at $z/D=1.50$ start to develop a shape similar to that at $x/D=2.40$ ($z/D=0.0$) due to the development of the vortex pair attached to the second row, while the effectiveness level at $z/D=0.80$ has increased to around 0.25. Further downstream the jets are gradually diluted and by $x/D=39.1$ the effectiveness field is almost two dimensional.

5.7 Concluding remarks

The computational procedure developed in the present study has been extensively validated against experimentally documented flow fields from various sources. From the overall performance of the procedure it can be concluded that it predicts successfully the rather complex flow field arising in the DHC single row and double row configurations and it can therefore be used as a design tool in engineering practice.

During the investigation however, it became apparent that the procedure failed to predict some details of the flow field. In the discussion which follows the areas and the reasons of failure are identified while chapter 6 outlines some possible ways of removing the deficiencies.

The computational procedure was validated against experimental data for laminar flows (for which there is no uncertainty about the calculation of the viscosity) and against experimentally documented turbulent flows.

The validation of the procedure for laminar flows was carried out and the results are reported in Bergeles et al (48). The main conclusion

from that validation were :

(i) the performance of the procedure in predicting an experimentally documented DHC flow for 30° injection under turbine conditions was very satisfactory;

(ii) for blowing at 90° the procedure indicated negative u velocities behind the hole which violates one of the simplifying assumptions embodied in the procedure. A special treatment of this region was developed within the framework of the semi-elliptic equations, which allowed predictions to be made for blowing rates up to $M=0.1$ but not beyond. Consequently a limitation of the procedure became apparent for normal injection.

The validation study for turbulent flow presented in this chapter revealed the following major problems :

- (i) existence of reverse flow region behind the hole;
- (ii) uncertainty of the velocity boundary condition over the hole, mainly for small boundary layer thicknesses; and
- (iii) directional character of the effective viscosity.

The first problem, that of the existence of reverse flow region behind the hole was tackled in the same way as for laminar flow. Nevertheless, it poses a limitation on the width of applicability of the numerical procedure; for normal injection the maximum blowing rate was $M=0.1$, while for injection at 30° $M=1.0$.

The second problem, that of the uncertainty in the entry profile of the secondary jet, was identified during the study when attempts were made to predict the present measurements of single row for $M=0.5$ and $\delta l/D=0.095$. A numerical exploration of the effect of the

jet velocity profile on the predicted flow field behind the hole revealed that the actual shape of the profile does have a significant effect 2-3 hole diameters behind the hole, the effect diminishing a few diameters further downstream. Consequently whilst for the major part of the theoretical study (where the employed boundary layer thicknesses $\delta l/D$ were greater than 0.120) the assumption of uniform exit velocity seems to be reasonably accurate, this is not the case for $\delta l/D < 0.095$ since it seems the jet velocity profile is highly distorted.

Fortunately, the uniform velocity assumption can still be applied even for thin boundary layers without significant loss of accuracy for the far downstream the hole flow field, $x/D > 10$.

The third problem arises from the necessarily approximate representation of turbulence by a turbulence model. Indeed a major conclusion of the present investigation is that the non-isotropic version of the (K- ϵ) model here developed proved to be reasonably successful in representing the turbulence structure of the DHC flow field. However, the model proved to be deficient on two occasions, namely :

- (i) at high blowing rates, for which the jet lifts off the surface, and
- (ii) for the double row DHC geometry, especially in the region between the two rows of holes.

The probable reason for the first deficiency has been already pointed out; the assumption that $\frac{\partial u_1}{\partial x_2}$ and $\frac{\partial u_1}{\partial x_3}$ are the only significant velocity gradients in the flow field. Apparently this assumption becomes invalid for high blowing rates because other velocity gradients become significant, i.e. $\frac{\partial u_2}{\partial x_3}$. Consequently the neglected velocity gradients

should be incorporated into the proposed non-isotropic (K- ϵ) model, however rendering the model more complicated.

A numerical exploration for the double row DHC case revealed that in the region between the two rows and downstream of the second row the diffusion coefficients tend to be nearly isotropic, while in the computations Γ_3/Γ_2 is assumed to be a function only of distance from the wall divided by boundary layer thickness. It seems, however, that the interaction of the jets from the two rows tends to equalize the lateral transverse fluctuations (i.e. $\overline{u_2^2} \approx \overline{u_3^2}$), and hence $\Gamma_3 \approx \Gamma_2$.

Further downstream of the second row of holes the relations Γ_3/Γ_2 reverts back to the non-isotropic form adopted throughout as the model for numerical computations.

In conclusion to the work presented in the chapter it can be stated that the developed procedure is :

- (i) flexible as regards boundary conditions and cooling geometry;
- (ii) economical; and
- (iii) reasonably reliable.

However there remain some improvements to be made.

CHAPTER 6

CONCLUDING REMARKS

6.1 Introduction

The purpose of this chapter is to summarise the work presented in this thesis, to assess the successes and the failures of the work and to outline areas which need further investigation.

6.2 A summary and assessment of the present investigation

The review of the previous studies of Chapter 1 revealed the absence of any prediction procedure for discrete hole cooling processes. Hence the main objective of the present work was to develop and validate such a prediction procedure. For the validation of the procedure however, there is need for experimentally documented flow fields, especially in the region near the injection hole. The review of the previous work however also showed that there was a lack of experimental data for the near-hole region; moreover nearly all the experimental studies were concerned with the wall film cooling effectiveness in the field. Hence in order to achieve the first objective, it was seen as necessary to take as a second objective of the study to investigate experimentally the near field region.

6.2.1 Experimental investigation

The detailed experimental measurements reported in Chapter 3, examined various discrete hole-cooling arrangements, i.e. injection through a single hole (90° and 30°) and single and double rows (30°) for various blowing rates and boundary layer thicknesses. The study

has shed light on various aspects of DHC systems which had not hitherto received attention.

The particular conclusions of each experimental study case were reported in Chapter 3 while a summary interlinking the drawn conclusions for the whole experimental study was presented at the end of Chapter 3, page 74.

6.2.2 Theoretical investigation

The extensive experimental study of the various DHC geometries was used in the validation of the prediction procedure. The theoretical work undertaken for the development of the procedure and the various contributions of the present study can be summarised as follows.

The major conclusion of the work presented in Chapter 5 was that the developed numerical procedure can predict very satisfactorily the hydrodynamic and temperature fields arising in practical situations of discrete-hole cooling processes. The development and the validation of such a prediction procedure was the main objective of the present work.

In particular the computational procedure predicted successfully a variety of experimentally documented flow fields for single hole normal and inclined injection and single row for a variety of blowing rates, hole spacings and density ratios (density of coolant to the mainstream). Finally the procedure was also tested successfully in predicting more complex flow fields, that of a double row cooling geometry and single row one under typical turbine blade operating conditions.

The predictive ability of the developed procedure depended crucially on how accurately

(i) the turbulence was mathematically represented by the turbulence model,

(ii) the mean flow field was represented by the approximated governing equations, and

(iii) the numerical solution procedure was solving the governing equations.

Below we assess in some more detail the achievements of the work on each of the above subjects.

(a) Theoretical contribution on the turbulence modelling

The main theoretical contribution of the present study was the proposal for approximating the Reynolds stresses and turbulent heat fluxes by a non-isotropic turbulence model. The non-isotropic ($k - \epsilon$) model adopted is expressed by equations (4.38), (4.39) and (4.36).

(b) Theoretical contribution on the governing equations

Firstly the equations governing the flow field were derived for a quasi-orthogonal co-ordinate system, which is particularly convenient for DHC calculations. The recommended form of the equations is expected to be accurate for rates of grid expansion up to 18° , above which the neglected terms become significant. These equations may be useful in predicting certain classes of flows, i.e. flow in rectangular diffusers.

Secondly an attempt was made to classify mathematically the practically important class of semi-elliptic flows. Despite the fact

that a mathematically necessary and sufficient condition for such a classification was not derived the present analysis does allow the distinction to be made between semi-elliptic or parabolic flows.

(c) Theoretical contribution on the numerical solution procedure

There are three notable contributions to the numerical solution procedure from the present study. The first one refers to the reverse flow region for which a special treatment was developed so it could be treated within the framework of the semi-elliptic procedure. The treatment was based on the assumptions that this region is small on extent and influence, assumptions which are valid for low blowing rates and low angles of injection. These restrictions however do pose limitations on the width of applicability of the procedure.

A second contribution was the development of a treatment for tackling free boundaries of the flow field in conjunction with the development of an entrainment law for the calculation of the rate of expansion of the free boundary.

Finally, a notable contribution of the present work was the development of a set of general computer codes for parabolic and semi-elliptic flows, named TEACH-3P and TEACH-3S respectively. These were based on the TEACH-3E code for 3-dimensional elliptic flows which had been developed earlier by Gosman-Humphrey and Vlachos.

During the validation of the procedure two main reasons emerged on why the predictions generated by the procedure were sometimes unsatisfactory. They are:

- (1) uncertainty in the jet velocity profile at the exit of the hole (this uncertainty is very crucial for boundary layer thickness $\delta_1/D < 0.09$),

- (2) the existence of regions of strong reverse flow.

These limitations on the procedure however can be removed by the methods outlined below.

6.3 Recommendations for future work

The areas recommended for further study are the following:

- (1) Improvement to the computational procedure;
- (2) Improvement to the turbulence modelling;
- (3) Application of the developed procedure to other DHC geometries
- (4) Extension of the experimental programme.

These topics will now be individually discussed.

6.3.1 Improvements to the computational procedure

(1) Calculation of the coolant velocity profile at the hole exit

As was reported earlier, the computational procedure failed to predict successfully the flow and concentration fields a few diameters behind the hole for conditions of small boundary layer thickness. The failure was attributed to the lack of knowledge of the coolant velocity profile at the hole exit. A method will now be outlined of calculating this velocity profile and so removing the above mentioned uncertainty.

The profile becomes highly distorted due to the pressure disturbances resulting from the interaction of the jet with the mainstream flow, an effect which cannot be simulated by a calculation procedure which requires the exit profile as input information. The new approach here proposed is to also calculate the flow field within the secondary pipe, coupling the external and internal fields through

the common pressure which prevails at the hole exit. The flow in the secondary pipe is itself semi-elliptic since the pressure field at the hole exit disturbs the flow upstream as was shown in the measurements described in Chapter 3. Consequently it is proposed the following procedure. Firstly an approximate solution of the external flow field would be obtained (i.e. by making a few iterations) for a guessed distribution of velocity over the hole. Then the semi-elliptic flow field in the secondary pipe would be calculated using as boundary conditions at the exit plane the pressure distribution from the calculation. This combined solution of both flow fields should give, after repeated applications convergence, the right velocity distribution at the hole exit.

It should however be noted that whilst for normal injection the solution on the secondary pipe is straightforward, some care must be taken for the inclined injection case to arrange that the marching procedure will have as its last calculation plane, the plate surface, which is now a plane inclined to the axis of the pipe.

(2) Treatment of the reverse flow region within the framework of a semi-elliptic procedure

The treatment proposed here which is the result of a discussion with A.D. Gosman (74), would be able to handle flows which are predominantly semi-elliptic but which include regions of recirculation. The main idea is based on allowing downstream effects to travel upstream through both convection and diffusion fluxes. (However, for simplicity and for the initial stages of the work, diffusion fluxes in the

streamwise direction could be ignored). The idea can be made practical by treating the reverse flow regions as fully elliptic and hence storing the hydrodynamic and temperature field quantities of this region in 3-dimensional arrays. The numerical consequence of the above proposals is that an extra "source" term representing downstream influences is introduced into the set of equations of the form

$$\begin{aligned} S_u &= - \frac{\left\{ \rho_u - |\rho_u| \right\}_P}{2} (\text{area})_d \phi_D \\ S_p &= \frac{\left\{ \rho_u - |\rho_u| \right\}_P}{2} (\text{area})_d \end{aligned} \quad (6.1)$$

One possible way the treatment is proposed to work is as follows:

- A preliminary run of a few iterations would be made with the standard semi-elliptic procedure for the purpose of determining the dimensions of the reverse-flow region,

- Next special 3-dimensional arrays would be created for the storage of all variables in this region and the difference equations for the cells therein would be modified as described above,

- The hybrid semi-elliptic/elliptic procedure would then be employed to solve for the flow field with possible readjustment of the boundaries of the elliptic region, if necessary.

The advantage of the proposed treatment over a fully elliptic one is the considerable saving of computer storage since the 3-dimensional storage employed is only for a small region of the flow field and not the entire field as is conventionally done.

(3) The rate of convergence of the procedure

A matter which needs urgent investigation is the rate of convergence of the semi-elliptic procedure. Despite the fact that the procedure has many advantages, for the problems for which it is intended, over a fully elliptic procedure there is considerable room for improving its economy, by reducing the number of iterations required. In this regard, there are three areas which need reappraisal.

(i) The assumption needs reconsideration that the changes in the pressures in a particular cross-stream plane affect only the streamwise velocities downstream of it and not the upstream ones since it is basically physically unrealistic. At least if, for computational reasons, the effect of the pressure changes on the upstream velocities cannot be taken into account, there should be developed a scheme which adjusts the entire upstream pressure field as each integration is visited.

(ii) The current practice of underrelaxing the pressure field and not the velocities should be reviewed. In fact, since the slowest converging of all the quantities is the pressure field, this suggests that the pressure field should not be underrelaxed but a form of underrelaxation should be imposed on the velocity field. It should be noted that it would be necessary to remove the underrelaxation with increasing number of iterations, otherwise it would influence the final solution.

(iii) The slow rate of convergence of the solution procedure is due mainly to the pressure field. Hence, this suggests that even though the use of the linearised momentum for correcting the velocity

field is adequate, the calculation of the pressure field from the addition on the prevailing pressure distribution of the calculated pressure corrections is not adequate. Consequently, another means is required for extracting a better pressure field distribution by using the corrected velocity field, or by solving a Poisson equation for the pressure itself and not for the pressure correction.

6.3.2 Improvements to the turbulence model

(1) Improving the proposed non-isotropic (k - ϵ) model

During the validation of the prediction procedure it emerged that the proposed non-isotropic turbulence model was not representing accurately the level of non-isotropy of the diffusion coefficients for circumstances of high blowing rates and double row of holes geometry.

The inadequacy of the model is attributed to the neglect of the velocity gradients, except those involving u velocity, in the derivation of the non-isotropic model and also possibly to the adopted simplified relation Γ_3/Γ_2 based on a fit to experimental data for pipe flow. It is likely that the model can be extended to cover these situations by incorporating into the analysis the neglected velocity gradients and employing a realistic modelling of the normal stresses $\overline{u'^2_3}$ and $\overline{u'^2_2}$ from which the ratio Γ_3/Γ_2 can be calculated (rather than the presented variation of Γ_3/Γ_2 adopted at present).

(2) A model for representing a turbulent jet in a laminar stream

In many practical situations, the flow field exhibits a mixed

laminar/turbulent character, i.e. the external flow over the blade is sometimes laminar but the jet issuing from the hole is fully turbulent. Such type of flows cannot be handled by the present form of the procedure since the turbulence model assumes fully turbulent mainstream and secondary flow.

Hence theoretical work should be undertaken for modifying the present $(k - \epsilon)$ model, so as to handle such mixed flows.

6.3.3 Future applications of the computational procedure

The present computational procedure can be used to examine in detail the influence of the various parameters on the cooling performance of various DHC configurations. In particular the following are suggested:

(1) An operational parametric study, i.e. determination of effects of α , M , dp/dx , T_u , ρ_c/ρ_∞ etc.

(2) A geometrical parametric study (effects of s/D , shape of holes, etc)

(3) For the double row of holes geometry the effect of the streamwise hole spacing, S_x/D , should be examined as well as the effect of having slits at locations between the two rows of holes in order to reduce the hot spots known to exist in these regions.

(4) A mode of injection whose behaviour was not examined in the present study was the effect on cooling performance of lateral injection. This arrangement in which the jets are directed laterally to the mainstream when viewed from above, so as to produce higher lateral spread, could be investigated with the computational procedure.

(5) Apart from the areas outlined above, the procedure could be employed in the prediction of the cooling, via discrete holes, of the discs of turbine blades (an area which urgently needs investigation) as well as the cooling of combustion chambers (particularly after suggestion 6.3.1 is introduced into the procedure).

(6) Throughout the present study, it was assumed that the blade surface was adiabatic. In practice, however, this is not the case because there is heat flux to the blade. The developed procedure is able to predict also these situations where there is non zero heat flux from the external stream to the blade.

6.3.4 Extension of the experimental study

On the basis of the work reviewed in Chapter 1 and that performed in the present study it can be suggested that the subject of discrete hole cooling for a single row of holes has been adequately covered by experiments. There are however a few areas which need experimental investigation, namely:

- (1) lateral injection, and
- (2) double row geometry.

For the first subject area there is no detailed experimental investigation (ref (27) is concerned with a single hole, while in ref (23) the hole spacing was very big so the individual jets were not interacting). As regards the double row geometry, this area needs more detailed investigation. In particular,

- (a) Measurements are needed to establish the optimum hole spacing (in the lateral and streamwise direction) for particular circumstances.
- (b) To examine the optimum hole arrangement (staggered or in line).
- (c) To explore an optimum arrangement of slits in the hot spot region of the rows and generally.

(d) To investigate optimum conditions for achieving maximum cooling effectiveness with minimum aerodynamic losses.

References

1. Fullagar, K.P.L. "The Design of air cooled rotor blades" Rolls-Royce (1971) Ltd., September 1973.
2. Bayley, F.J. and Turner, A.B. "Transpiration cooled turbines" AGard Conference Proceedings, Paper 12, High temperature turbines, AGard-CP-73-71.
3. Helms, H.E. and Emmerson, C.W. "Analysis and Testing of air cooled turbine rotor and stator blades" AGard Conference Proceedings, No. 9, Gas Turbines, 1966.
4. Eltis, E.M. and Wilde, G.L. "The Rolls-Royce RB211 Turbofan Engine" The Institution of Mechanical Engineers, Proceedings 1974, Vol. 188, 37/74.
5. Platten, J.L. and Keffer, J.F. "Entrainment in deflected axisymmetric jets at various angles to the stream" University of Toronto, TP6808, June 1968.
6. Keffer, J.F. and Baines, W.D. "The round turbulent jet in a cross-wind" J. Fluid Mech, 15, pp.481-496.
7. Margason, R.J. "The path of a jet directed at large angles to a subsonic free-stream" NASA TND-4919.
8. Chan, D.T. "Turbulent non-buoyant or buoyant jets discharged into flowing or quiescent fluids" Ph.D. thesis, Iowa University, 1972.
9. Vogler, D.R. "Surface pressure distributions induced on a flat plate by a cold air jet issuing perpendicularly from the plate and normally to a low-speed free stream flow" NASA TN D-1629, 1963.
10. Bradbury, L.J.S. and Wood, M.N. "The static pressure distribution around a circular jet exhausting normally from a plane wall into an airstream" A.R.C. CP No. 822.
11. Goldstein, R.J. "Film Cooling" Advances in heat transfer, Vol. 7, Edit by T.F. Irvine and J.P. Hartnett, 1971.
12. Wieghardt, K. "Hot air discharge for De-icing" AAF Trans. No. F-TS-919-RE, 1946.
13. Spalding, D.B. "Prediction of adiabatic wall temperatures in the film cooling systems" A.I.A.A Vol. 3, 965, 1965.
14. Eckert, E.R.G. "Film cooling with injection through holes" AGard Conference Proceedings, High Temperature turbines, AGard CP-73-71.
15. Eckert, E.R.G. and Drake, R.M. "Heat and Mass Transfer" McGraw Hill, 1959.
16. Mayle, R.E. and Camarate, F.J. "Multihole cooling effectiveness and heat transfer" A.I.A.A. ASME 1974, Thermophysics and Heat Transfer Conference.
17. Eriksen, V.L. Eckert, E.R.G. and Goldstein, R.J. "A model for analysis of the temperature field downstream of a heated jet into an isothermal cross flow at an angle of 90°" NASA CR-72990, 1971.

18. Pedersen, D. "Effect of density ratio on film cooling effectiveness for injection through a row of holes and for a porous slot" University of Minnesota, Ph.D. Thesis, 1972, Mech. Eng.
19. Kadotani, K. "Effect of mainstream variables on heated and unheated jets issuing from a row of inclined round holes" University of Minnesota, Mech. Eng, Ph.D. Thesis, Dec. 1975.
20. Liess, C. "Film cooling with ejection from a row of inclined circular holes" Von Karman Institute, TN97, March 1973.
21. Liess, C. and Carnel, J. "Application of film cooling to gas turbine blades" AGard Conference Proceedings, No. 73 on high temperature turbines, 1971.
22. Launder, B.E. and York, J. "Discrete-hole cooling in the presence of free-stream turbulence and strong favourable pressure gradient" Int. J. Heat Mass Transfer, Vol. 17, Pl403, 1974.
23. Le Brocq, P.V. Launder, B.E. Priddin, C.H. "Discrete-hole injection as a means of transpiration cooling; an experimental study" Institution of Mechanical Engineers, Proceedings, vol. 187, 17/73, 1973.
24. Smith, M.R., Jones, T.V. and Schultz, D.L. "Film cooling effectiveness from rows of holes under simulated gas turbine conditions" A.R.C. No. 1303, 1974.
25. Jabbari, M.Y. "Film cooling and heat transfer with air injection through a staggered row of holes into an accelerating flow" University of Minnesota, Ph.D. Thesis, 1973.
26. Goldstein, R.J., Eckert, E.R.G. and Ramsey, J.W. "Film cooling with injection through holes: Adiabatic wall temperature downstream of a circular hole" Transactions of the ASME, p384, October 1968.
27. Goldstein, R.J., Eckert, E.R.G., Eriksen, V.L. and Ramsey, J.W. "Film cooling following injection through inclined circular tubes" Israel Journal of Technology, Vol. 8, No. 1-2, p. 145, 1970.
28. Goldstein, R.J., Eckert, E.R.G., Eriksen, V.L. and Ramsey, J.W. "Film cooling following injection through inclined circular tubes" NASA CR-72612, 1970.
29. Eriksen, V.L. "Film cooling effectiveness and heat transfer with injection through holes" NASA CR-72991, 1971.
30. Goldstein, R.J., Eckert, E.R.G. and Burggraff, F. "Effects of hole geometry and density on three-dimensional film cooling" Int. J. Heat Mass Transfer, Vol. 17, p.595, 1974.
31. Ramsey, J.W. and Goldstein, R.J. "Interaction of a heated jet with a deflecting stream" NASA CR-72613.
32. Krusse, H. "Film cooling measurements" Internal report 352-74/9, DFVLR, September 1974.

33. Brown, A. "Private communication" Mech. Eng. Dept., University of Cardiff, 1975.
34. Metzger, D.E., Takenchi, D.I., Krentler, D.A. "Effectiveness and heat transfer with full coverage film cooling" ASME paper, No. 73-GT-18, April 1973.
35. Lander, R.D., Fish, R.W. and Suo, M. "External heat-transfer distributions on film cooled turbine vanes" J. of Aircraft, Vol. 9, 1972, pp. 107.
36. Baker, R.J. and Launder, B.E. "The turbulent boundary layer with foreign gas injection II
Int. J. Heat Mass Transfer, Vol. 17, No. 2, 1974.
37. Bayley, F.J. Morris, W.D., Owen, J.M. and Turner, A.M. "Boundary-layer prediction methods applied to cooling problems in the gas turbine" ARC C.P. No. 1164, 1971.
38. McGuirk, J. "Prediction of turbulent buoyant jets in co-flowing streams" Ph.D Thesis, Imperial College, Dept. Mech. Eng. May 1975.
39. Patankar, S.V., Rastogi, A.K. and Whitelaw, J.H. "The effectiveness of three-dimensional film cooling slots - Predictions" Int. J. Heat Mass Transfer, 16, p.1673, 1973.
40. Bradshaw, P. "Experimental fluid mechanics" Pergamon Press, 1970.
41. Ribeiro, M.M. "The turbulence structure of free jet flows with and without swirl" Imperial College, Mech. Eng. Dept, Ph.D. Thesis, 1976.
42. Rastogi, A.K. "Effectiveness and heat transfer downstream of three-dimensional film cooling slots" Imperial College, Mech. Eng. Dept, Ph.D. Thesis, 1972.
43. Baker, R.J. "The turbulent boundary layer with foreign gas injection" Imperial College, Mech. Eng. Dept, Ph.D. Thesis, 1972.
44. Ribeiro, M.M. and Whitelaw, J.H. "Verification of turbulence models with hot wire anemometry" Imperial College, Mech. Eng. Dept., report HTS/72/20, 1972.
45. Bergeles, G., Gosman, A.D., Launder, B.E. "The near-field character of a jet discharged through a wall at 90° to a mainstream" ASME Winter Annual Meeting, 75-WA/HT-108, 1975.
46. Margason, R.J. and Fearn, R. "Jet wake characteristics and their induced aerodynamic effects on V STOL aircraft in transition flight" NASA SP-218, 1969, p.1.
47. McMahon, H.M. and Mosher, D.K. "Experimental investigation of pressures induced on a flat plate by a jet issuing into a subsonic stream" NASA SP-218, 1969, p.49.
48. Bergeles, G., Gosman, A.D., and Launder B.E. "The prediction of the three-dimensional cooling processes : I. Laminar flow" ASME Winter Annual Meeting, 1975, 75-WA/HT-109.

49. Batchelor, G. K. "An introduction to Fluid Dynamics", Cambridge University Press, 1974.
50. Goldstein, S. (Editor) "Modern developments in Fluid Mechanics", Dover Publications, Inc., 1965.
51. Bradshaw, P. "An introduction to Turbulence and its Measurements", Pergamon Press, 1971.
52. Hinze, J. O. "Turbulence", McGraw-Hill Book Co. Ltd., N. Y.
53. Rodi, W. "The prediction of free turbulent boundary layers by use of a 2-equation model of turbulence", Imperial College, Mechanical Engineering Department, Ph.D. thesis, 1972.
54. Launder, B. E. "An improved algebraic modelling of the Reynolds stresses" Imperial College", Heat Transfer Section report, TM/TN/A9, April 1971.
55. Launder, B. E., Reece, G. and Rodi, W. "Progress in the development of a Reynolds Stress turbulence closure", J. Fluid Mech., Vol. 68, pp. 537 - 566, 1975.
56. Launder, B. E. "The physical modelling of heat and momentum transport in buoyancy affected turbulence", Imperial College, Mechanical Engineering Department report, February 1974.
57. Black, A. W. and Sparrow, E. M.
J. Heat Transfer, 89c, 258, 1967.
58. Quarnby, A. and Quirk, R. "Axisymmetric and non-axisymmetric turbulent diffusion in a plain circular tube at high Schmidt number", Int. J. Heat Mass Transfer, 143, 17, 1974.
59. Quarnby, A. and Quirk, R. "Measurements of the radial and tangential eddy diffusivities of heat and mass in turbulent flow in a plain tube", Int. J. Heat Mass Transfer, 15, 2309, 1972.
60. Launder, B. E. "Topics in applied physics turbulence", Edited by P. Bradshaw, Springer, 1976.
61. Launder, B. E. and Spalding, D. B. "Mathematical models of turbulence", Pergamon Press, 1972.
62. Jones, W. P. and Launder, B. E. "The prediction of laminarisation with a 2-equation model of turbulence", International J. Heat and Mass Transfer 15, 301.
63. Spalding, D. B. "Numerical methods in fluid mechanics and heat transfer", Imperial College, Dec. 1970, EF/TN/A/34.
64. Antonopoulos, K. "A comparison of the finite-element and finite difference method for the solution of the diffusion-convection equation", Imperial College, M.Sc. Thesis, 1974.
65. Patankar, S. V. "On available calculation procedures for steady, three-di-

- mensional boundary layers", Imperial College, Dept. Mechanical Engineering, Rep. BL/TN/A/44, 1971.
66. Patankar, S. V. and Spalding, D. B. "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows", Int. J. Heat Mass Transfer, 15, (10), pp. 1787.
 67. Chorin, A. J. "Numerical solution of the Navier-Stokes equations", Math. Computation, 22, p. 745, 1968.
 68. Gosman, A. D. "Lecture 4, A finite-difference calculation procedure for turbulent flows", Imperial College, M.Sc. course, 1973.
 69. Gosman, A. D., Pun, W. M., Runchal, A. K., Spalding, D. B. and Wolfshtein, M. "Heat and mass transfer in recirculating flows", Academic Press, London - New York, 1969.
 70. Pratap, S. V. "Flow and heat transfer in curved ducts", Imperial College, Dept. Mechanical Engineering, Ph.D thesis, August 1975.
 71. Launder, B. E. and Spalding, D. B. "The Numerical computation of turbulent flow", Computer Methods in Applied Mechanics and Engineering, Vol. 3, 1974, pp. 269 - 289.
 72. Tatchell, D. G. "Convection processes in confined three-dimensional boundary layers", Imperial College, Mechanical Engineering Dept., Ph.D Thesis, July 1975.
 73. Gosman, A. D., Simitovic, R. "Private Communication"
 74. Gosman, A.D. "Private Communication".
 75. Pankhurst, R.C. and Holder, D.W. "Wind tunnel technique", Pitman, 1952.
 76. Gosman, A.D. "Private Communication".
 77. Bergeles, G., Gosman, A.D. and Launder, B.E. "The prediction of discrete-hole cooling. 1. A progress report on experiences with parabolic and semi-elliptic versions of STABL". Imperial College, Mech. Eng. Dept. Report, August 1974.

NOMENCLATURE

C_p	pressure coefficient $C_p = (P - P_{ref}) / (\rho_j u_\infty v_j / 2)$
c_p	specific heat at constant pressure
C	concentration of tracer gas or non-dimensionalised temperature
C_D	coefficient in the turbulence model, $C_D = 0.09$
D	hole diameter
e_{ij}	rate of strain tensor
f	weighting factor
k	turbulent energy ($k = \frac{1}{2} u_i'^2$)
h	total enthalpy
M	injection ratio ($M = \rho_j v_j / \rho_\infty u_\infty$)
P	static pressure
Pe	Peclet number
Re	Reynolds number ($Re_d = \frac{u_\infty D}{\nu_\infty}$)
S	lateral hole spacing
S_x	streamwise row spacing
T	temperature of fluid
$\bar{T}$	non-dimensionalised temperature ($\bar{T} = \frac{T - T_\infty}{T_j - T_\infty}$)
T_u	turbulence intensity ($T_u = \frac{k}{u}$)
t	time
u_T	friction velocity
u, u_1	x, x_1 direction velocity (streamwise)
v, u_2	y, x_2 direction velocity
v_j	mean resultant injection velocity
$\bar{v}_j$	mean injection velocity
w, u_3	z, x_3 direction velocity
x, x_1	streamwise direction ($x=0$ at the upstream edge of the upstream row of holes)
y, x_2	normal to the surface direction ($y=0$ at the surface)
y^+	defined as $y^+ = \rho u_T y / \mu$
z, x_3	lateral direction ($z=0$ at the centre line bisecting a hole of the upstream row of holes)

GREEK SYMBOLS

α	angle of injection
Γ_{Φ}	diffusion coefficient for Φ variable
δ	boundary layer thickness
δ_1	displacement thickness
ϵ	dissipation rate of k
η	wall film cooling effectiveness $(\eta = (c_p T_w - c_{p_{\infty}} T_{\infty}) / (c_{p_j} T_j - c_{p_{\infty}} T_{\infty}))$
$\bar{\eta}$	spanwise average wall effectiveness
θ	angle of grid expansion
μ	laminar dynamic viscosity
ρ	density
σ	Prandtl number
τ	Shear stress
(ξ, η, ζ)	curvilinear co-ordinate system
ξ	streamwise direction
η	normal to the surface direction ($0 \leq \eta \leq 1$)
ζ	lateral direction

SUBSCRIPTS

CL	centre line
off CL	line parallel to the CL
f	fluid
ref	reference point
tot	total
inj	injection
j	conditions at the hole exit
w	conditions at the wall
∞	conditions at the free stream
c	coolant
2	x_2 direction or y direction
3	x_3 direction or z direction
x	x direction
y	y direction

Appendix A

Error analysis of the velocity measurements downstream of the injection hole using total and static pressure probes

A1 Introductory remarks

The errors occurring during the measurements of the velocity profiles using a pitot and a static tube can be attributed to the following:

- (1) uncertainties in the flow rates;
- (2) inaccuracies in the measuring probe;
- (3) inaccuracies in the recording apparatus;
- (4) the experimentalist.

Below is estimated the error due to each of the above factors and from these the total error in the measured near-wall streamwise velocity. The measurement location chosen for this analysis is the one nearest and behind the hole, where it is expected that the maximum error will occur.

A2 Error due to determination of the flow rates

The secondary flow rate was measured by an orifice plate. The pressure drop across the orifice was kept constant to within 2% of the accurate value during the measurements (suggesting that the secondary flow rate was kept constant to within 1%). It is assumed that the influence on the total head and static pressure of the flow at the location considered behind the injection hole is of the same order of magnitude.

The change of the total head of the free stream was changing within 1% of the nominal "wind tunnel" value during the experimental study.

A3 Error due to Measuring instrument

There are numerous possible sources of error, due mainly to:

- (i) Non-alignment of the probe with the flow;
- (ii) viscous effects;
- (iii) high turbulence levels;

For each of the above cases the estimated error is as follows:

A3.1 Non-alignment of the probe with the flow

The inaccuracies in the estimated total and static pressures are functions of the yaw angle between the probe and the main flow, displayed in Figure A1 (taken from (75)). The non-alignment of the probe with the flow is estimated from the corresponding computer predictions for these cases and the angles so determined, as well the calculated turbulence levels are tabulated in table A1 for various cases.

Of course the predictions are themselves subject to errors; however, this was the only feasible means to proceed.

A3.2 Viscous effects

According to reference (75) the viscous effects on the measurements of total and static pressure depend on the local Reynolds number Re_d (defined with characteristic length the diameter of the probe) and these effects are negligible for $Re_d > 10$. In all the present measurements $Re_d > 200$, then viscous effects are unimportant.

A3.3 High turbulence levels

High turbulence levels cause the probes to overestimate the total pressure, but there is no significant effect on the measurement of static pressure, according to ref (75). Ref (75) gives the relation between the non-dimensional total pressure and the turbulence level as:

$$\frac{P_{tot} - p}{\frac{1}{2} u^2} = 1 + \frac{k}{u^2} \quad (A1)$$

where p is the static pressure,

P_{tot} is the total pressure

k is the kinetic energy of turbulence.

The turbulence level immediately downstream of the hole is very high and according to Table A1, it is this source of error which makes the most significant contribution to the velocity inaccuracy.

A3.4 Presence of the wall

Due to the presence of the wall, the probe reading is greater than the total pressure of the undisturbed flow at the geometrical centre of the tube. As explained in ref (75) this error can be compensated for by an outward displacement of the effective centre of the probe for the point nearest to the wall by an amount

$$\Delta y = 0.15d \quad (A2)$$

A3.5 The recording instrument

The recording instrument used for the total and static pressure was a digital differential pressure anemometer capable of measuring up to 10^{-3} mm WG. The high sensitivity of the instrument makes the uncertainties due to the recording instrument generally negligible.

A3.6 The experimentalist

The pressure was recorded by the instrument to three digits. Just behind the hole the reading of the digital manometer was fluctuating and it is estimated that an error of 3% in the mean value was resulted from this.

In the analysis and accompanying table below the errors attributed to each source are summarised and the total maximum error in the measurement of the near-wall velocity is calculated for each case. The calculations are based on the following relations

$$\text{Total error} = \sqrt{\left\{ (\text{Partial error of source 1})^2 + (\text{partial error of source 2})^2 + \dots \right\}} \quad (A3)$$

The velocity is calculated from the relation

$$u \propto \sqrt{P_{\text{tot}} - P} \quad (A4)$$

and the error in u is equal to

$$\text{Fractional error in } u = \sqrt{1 + \frac{(\text{Error in } P_{\text{tot}} + \text{Error in } P)}{P_{\text{tot}} - P}} \quad (A5)$$

(P_{tot} = total head, P = static pressure)

A.4 The experimental uncertainty of the measurements

Table 1 below summarises the errors in the measurements due to various sources.

1 case single hole	2 x/D	3 estimated yaw angle	error due to yaw		5 turbulence level k/u ²	6 error due to k on P _{tot}	7 fluctuat. of flow rate	8 error on u	9 error due to experim P _{tot}	10 as 9 P	11 12 total error in		13 error in u due to P _{tot}	14 TOTAL ERROR IN u
			P _{tot}	P							P _{tot}	P		
α = 90° M = 0.046	1.4	4	0	0	0.09	0.09	0.02	0.02	0.03	0.03	0.092	0.03	0.018	0.021
α = 90° M = 0.1	2.5	15	0	0	0.13	0.13	0.02	0.02	0.03	0.03	0.132	0.03	0.04	0.041
α = 30° M = 0.1	2.5	10	0.005	0.025	0.21	0.21	0.02	0.02	0.03	0.03	0.212	0.039	0.059	0.05
α = 30° M = 0.2	2.5	12	0.008	0.03	0.30	0.30	0.02	0.02	0.03	0.03	0.302	0.044	0.19	0.19
α = 30° M = 0.3	2.5	18	0.04	0.07	0.35	0.35	0.02	0.02	0.03	0.03	0.354	0.077	1.35	1.35

Table A1: Estimation of the experimental errors

For blowing rates higher than M = 0.30 there is reverse flow behind the hole, hence the estimated uncertainty is very high for this case. It should be mentioned that however the measurements at these locations of high uncertainty were at least reproducible to within 5-10%.

A.5 Error analysis of the wall film cooling effectiveness measurements

The wall film cooling effectiveness was deduced from the measurements of the concentration of the tracer gas in the flow and in the secondary pipe using the relation

$$\eta = C_w / C_{inj} \quad (A6)$$

From the above relation we can deduce an estimate of the accuracy of the wall effectiveness

$$\frac{\Delta \eta}{\eta} = \sqrt{\left\{ \frac{\Delta C_w}{C_w} \right\}^2 + \left\{ \frac{\Delta C_j}{C_j} \right\}^2} \quad (A7)$$

where C_w and C_j due to the fluctuations mainly of secondary flow rate

* The experiments show a reverse flow region behind the hole at the location of measurements.

. sampling rate (fluctuation of the pump vacuum)

Inserting in relation (A.7) typical values of C_w , C_j , ΔC_w and ΔC_j we deduce that the accuracy of the effectiveness depends on its level and in particular for $\eta > 80\%$ the inaccuracy is 2%, while for $\eta < 5\%$ the inaccuracy on η rises to around 20%.

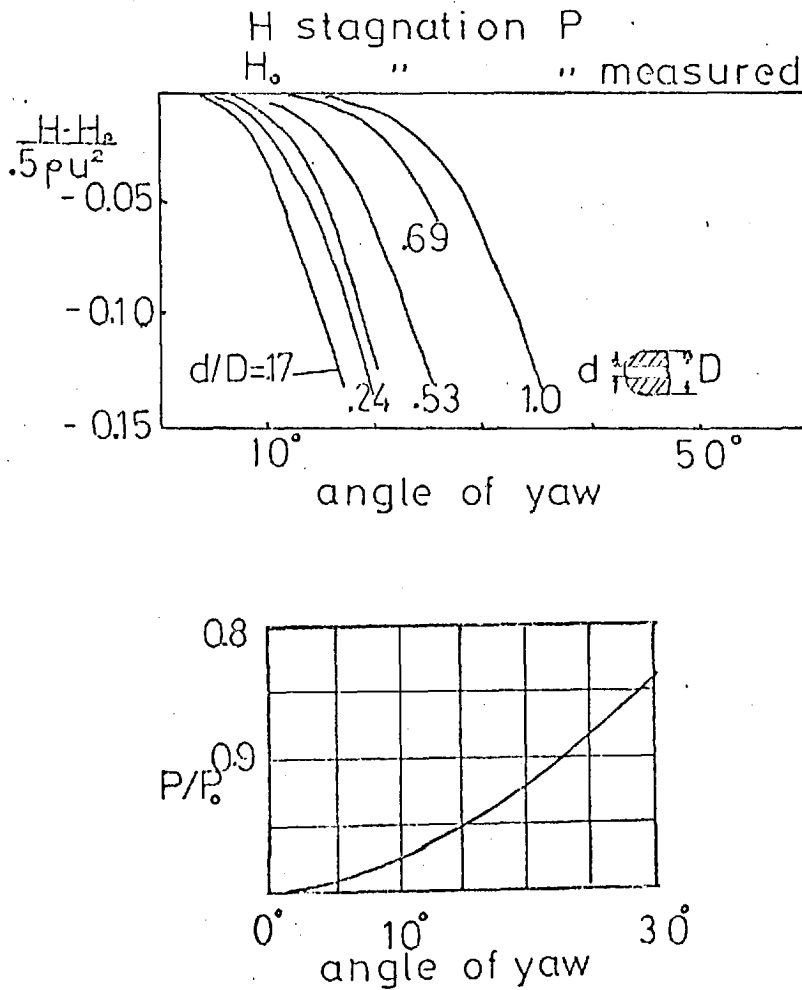


Fig. A1 : Influence of angle of yaw on total and static pressure /taken from (75)/.

Appendix B

Some detailed features of the computational procedure

B.1 Introductory remarks

In Appendix B we describe some novel aspects of the developed computational procedure; namely:

- (1) the treatment of reverse flow within the framework of semi elliptic transport equations;
- (2) the treatment of the free boundary;
- (3) the entrainment law

B.2 The treatment of reverse flow

B.2.1 The need for a treatment of the reverse flow region

One of the main assumptions by which the Navier-Stokes equations are transformed to parabolic form is that there is a predominant flow direction along which no flow reversal occurs. For the particular problem examined in this study the numerical procedure predicts reverse flow in the supposed predominant direction, thereby contradicting the foregoing assumption. Since in this case the flow field become in fact elliptic and should be treated as such or procedure for solving the parabolic equations should if possible either be altered to accommodate such regions.

B.2.2 Description of the mechanism which causes the failure of the semi-elliptic procedure

In the first stages of the work the treatment adopted for the reverse flow region was to simply replace the negative velocities by small near-zero positive values; this practice however did not suffice as it led to divergence of the semi-elliptic procedure. Explorations to determine the reasons for the divergence revealed the following: In the computational cells located within the region of reverse flow and for which the above practice was adopted, high mass imbalances were calculated (typical values

10-20 times greater than mass imbalances in the cells outside the reverse flow region). These high cell mass imbalances provoked through the pressure correction equation, high pressure corrections and consequently unrealistic lateral velocities. The distorted cross-stream velocity field affected the calculation procedure but not so as to cause divergence. The divergence of the procedure comes through the pressure field which, when used in the next iteration of the semi-elliptic procedure amplifies the distortion of the pressure and so on.

B.2.3 The cure

The treatment adopted was based on the knowledge that the reverse flow region is small in dimensions and weak in strength: consequently it can be replaced by an essentially stagnant region. Since the divergence of the numerical procedure was caused by the high residual mass sources in the cell in this region, it was thought that the reduction of these sources would be the solution to the problem.

One solution to the problem might be to insert in the pressure correction equation the calculated negative velocities: this is not however a certain cure because the velocities are themselves wrong, since there is no link with the downstream planes in the u momentum equation. This procedure was not therefore tried: instead the assumption was made that in these cells local continuity was satisfied and thus cell mass imbalances $\dot{m}_p^* = 0$, equation (4.63). The assumption can be justified if the reverse flow region is small and weak.

The incorporation of the above modification prevented divergence but still did not bring complete convergence since the sum of the absolute values of all the residual mass sources did not reduce below a level of 5% of the nominal flow rate.

B.2.4 Extension of the assumption of cell mass balance

A closer examination of the causes of non-convergence of the above practice revealed that high pressure corrections were again calculated, but now in the downstream cells, adjacent to the region of reverse flow. Obviously this should be expected because in these cells we use an upstream mainstream velocity which is wrong (because the negative u-velocity was set to zero). Extension of the specific treatment to the downstream cells (that is $m_p^* = 0$) brought complete convergence of the solution procedure.

B.3 The treatment of the free boundary

At the early stages of the calculations the entrainment v_∞ velocity at the edge of the boundary layer was calculated by solving finite difference form of the v momentum equation for the cell located at the free boundary. Whilst this treatment was satisfactory and gave correct results (compared with existing analytical solutions for laminar and turbulent flow over flat plate and laminar flow over a flat plate with suction) required considerable modification to the solution procedure for the v momentum equation. At a later stage in the work the above treatment was replaced by a substantially simplified one proposed by Gosman (76), which consisted of not modifying the existing treatment of the v momentum equation (consequently not solving for v_∞ but assuming $((\frac{\partial v}{\partial y})_\infty = 0)$ and taking that, the gradient $(\frac{\partial p}{\partial y})_\infty$ is also zero, hence $p'_{(N-1)} = 0$ (because p_∞ is specified).

The proposed treatment of the free boundary produces identical results with the earlier adopted one but it is far superior due to its simplicity in its application into the computational procedure.

B.4 The entrainment law

The rate of expansion of the computational grid in the y direction (and correspondingly in the z direction) was calculated by a form of entrainment law. The expansion formula derived by a momentum integral balance in the streamwise direction for a control volume located at the

free boundary as shown in figure B1.

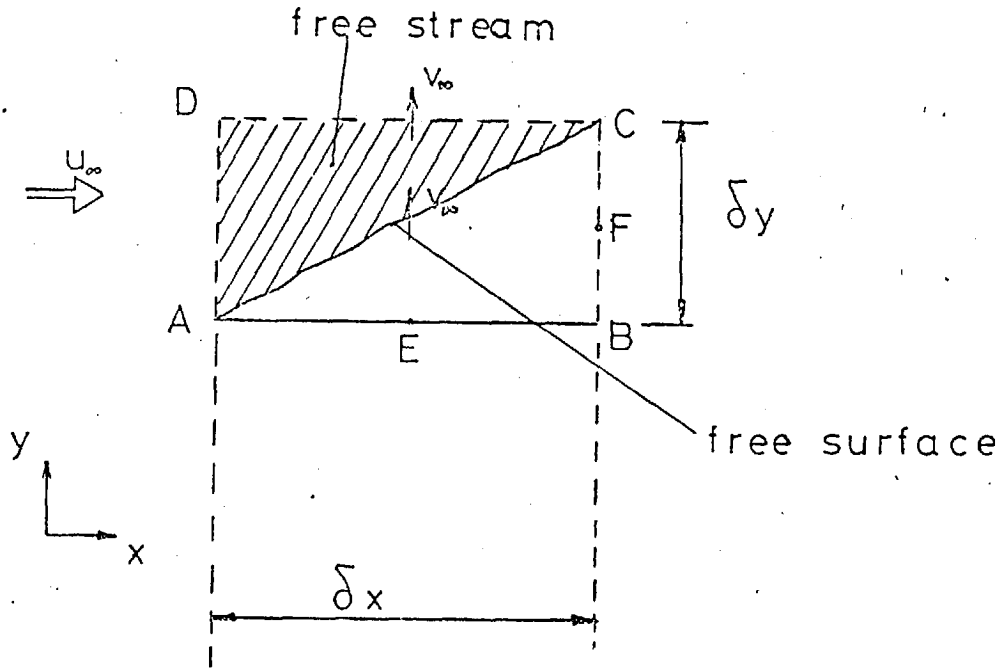


Fig. B1: Control volume for the entrainment law

The momentum balance may be written:

$$-\rho v_E u_E \delta x + \rho v_\infty u_\infty \delta y - \rho v_\infty u_\infty \delta y + \rho u_F u_F \delta y = -\mu \left(\frac{\partial u}{\partial y} \right) \delta x$$

$$\delta y \rho u_\infty^2 (1 - u_F^2/u_\infty^2) = u_\infty \delta x \left\{ \rho v_\infty - \rho v_E \frac{u_E}{u_\infty} + \frac{\mu (\partial u / \partial y)_E}{u_\infty} \right\}$$

$$\begin{aligned} \frac{\delta y}{\delta x} &= \left\{ \rho v_\infty - \rho v_E \frac{u_E}{u_\infty} + \frac{\mu (\partial u / \partial y)_E}{u_\infty} \right\} \left\{ 1 + \frac{u_F}{u_\infty} \right\}^{-1} \left\{ 1 - \frac{u_F}{u_\infty} \right\}^{-1} / \rho u_\infty \\ &= \left\{ \rho v_\infty \left(1 - \frac{v_E}{v_\infty} \frac{u_E}{u_\infty} \right) \left(1 - \frac{u_F}{u_\infty} \right)^{-1} + \frac{\mu (\partial u / \partial y)_E}{u_\infty \left(1 - \frac{v_F}{v_\infty} \right)} \right\} / \rho u_\infty \left(1 + \frac{v_F}{v_\infty} \right) \end{aligned}$$

Assuming that:

$$\frac{v_E}{v_\infty} \approx 1, \quad u_F \approx u_E \text{ and } 1 - \frac{u_F}{u_\infty} \approx 0.1$$

We finally get the rate of expansion of the grid in the y direction as:

$$\frac{\delta y}{\delta x} = \left(\rho v_\infty + \frac{\mu (\partial u / \partial y)_E}{0.1 u_\infty} \right) / 1.9 \rho u_\infty$$

Appendix C

The prediction of discrete-hole cooling with the parabolic computational procedure

C.1 Introductory remarks

It was mentioned in chapter 4 that early attempts to treat the near-hole region as parabolic failed because the solutions were physically unrealistic. The reason for the failure of the parabolic treatment was traced to the assumptions regarding the streamwise pressure gradient. It is an inevitable consequence of a strictly parabolic treatment that the streamwise pressure gradient must be uniform over the cross-section, with its value either prescribed as a function of x (for external flows) or calculated from the overall continuity requirement (internal flows). Below is summarised the various attempts to specify the streamwise pressure gradient and the solutions generated therefrom. For more details on our experiences using a parabolic and "semi-elliptic" versions of the developed numerical scheme, see Bergeles et al (77).

C.2 Specification of the streamwise pressure gradient - flow behaviour

C.2.1 External flow prescribed pressure gradient ($\alpha = 90^\circ$)

When the pressure gradient was prescribed as being everywhere zero, the computed flow behaviour was both unrealistic and, in some cases unstable. In particular, at the start of injection hole large normal velocities were predicted across the whole height of the boundary layer and these subsequently caused divergence in some instances. This practice was therefore discarded.

C.2.2 Application of a confined-flow procedure ($\alpha = 90^\circ$)

In order to suppress the large normal velocities, the flow over the hole was treated as "confined". The method was to locate the outer boundary far from the wall, where undisturbed two-dimensional flow could be expected to prevail. This boundary was represented as a frictionless porous wall, with a prescribed suction velocity through it comparable with the value of v_∞ .

predicted from 2D boundary-layer theory. The usual confined-flow procedure was then used to determine dp/dx considering either the whole cross-section or simply that portion of it occupied by fluid that had emanated from the jet (determined by a particle-path method). The pressure gradient was then applied either over the whole or a part of the cross section as appropriate*. The above practice indeed reduced the magnitude of the normal velocities but still the solutions exhibited implausible features. At every station over an injection hole the pressure-gradient correction dp/dx was negative and hence, the fluid to which the correction was applied was accelerated. In practice, however, over the injection zone, the streamwise velocity close to the wall is very low because the jet fluid enters with zero streamwise momentum. The solutions also exhibited a faster rate of lateral dispersion than anticipated, the wall concentration along the centre line remote from the injection hole reaching its maximum value before the end of the hole (i.e. at $x/D > 1$).

C.23 Local pressure gradient from streamline curvature

The above experiences led to the conclusion that the assumption of a uniform $\partial p/\partial x$ across the cross-section must be abandoned; in some regions of flow we needed $\partial p/\partial x$ taking both negative and positive values in the same plane.

Our first practice was to make an a posteriori estimate of $\partial p/\partial x$ by assuming radial equilibrium normal to the streamlines (the streamline curvature being estimated from the stored values of velocities at the two calculation planes immediately upstream). This practice brought some improvement over that described in C.22; at least regions of both positive

* The reason for applying the streamwise pressure gradient to just a portion of the cross section was that the predicted behaviour then became virtually independent of the arbitrary choice of the position of the imaginary top wall. Moreover the steepest streamline curvature (which is associated with steep pressure gradients) occurs near the hole.

and negative pressure gradient were present. However, because the pressure gradient was calculated from the curvature of upstream streamlines, the method could not produce the smooth deflection of the mainstream around the jet depicted in figure **c1**. A further cause of dissatisfaction was the high lateral rate of spread that was still present. From figure **c2** it is seen that the peak concentration along $z = S/2$ (i.e. the plane furthest from the injection hole) occurs at $x/D=1$. By contrast the "point-source" model outlined in chapter 1 predicts the off-centre-line peak to occur at about 5 diameters downstream. Now, although the point-source model is not an exact theory, for laminar flow its predictions should lie within a few tens of per cent of the exact solution.

C.2.4 Upstream pressure field from potential flow theory

Although real flows are not inviscid, the direct effects of viscosity on the pressure field are arguably weak. For example, in regions where the effective viscosity is uniform the Poisson equation for pressure, obtained by taking the divergence of the momentum equations, does not contain any viscous terms at all. Thus the possibility arises of using a solution to the simpler inviscid equations to provide values of the pressure over some of the flow. The region where such an approach would seem especially beneficial is that upstream of the point of injection. In fact, potential flow theory yields a simple analytical solution for the interaction of a uniform-velocity field with a point source.

As the sketches in figure **C1** suggest, the streamline pattern in this flow appeared much the same as in the discrete-jet upstream of injection. We therefore made some computations in which the practice for finding $\partial p / \partial x$ described in C.2.3 above was replaced by the pressure field given by the

the potential flow solution for the region upstream of an injection hole. This had the desired effect of causing the streamlines to deflect ahead of the discharge. There was however, the problem of relation u_{pot} to u_{∞} . A value of $u_{\text{pot}} = 0.5 u_{\infty}$ produced such severe pressure gradients near the wall as to provoke separation in front of the jet but too weak values of $\partial p / \partial x$ further from the surface. No single value gave satisfactory effects both near to and some distance remote from the wall. Evidently, despite the encouraging appearance, the flow field resulting from the point-source uniform-flow superposition did not, in fact, provide a useful approximation of the time pressure field. Further exploration of this approach was not pursued because it was felt that a semi-elliptic treatment of the flow field was most appropriate.

C. 3 Some characteristic results using the semi-elliptic procedure.

The flow field which was attempted to be calculated with the parabolic procedure was studied with the semi-elliptic one.

Perhaps the most striking evidence of the need for an elliptic treatment is provided by figure C3. It shows the computed variation with x of the static pressure along the wall for $z=0$ and for $z=S/2$, that is along the edges of the computational domain. Pressure differences are normalised by the dynamic head of the jet stream. Not only is there substantial pressure variation upstream of the hole; the computed variation of p over the hole itself and downstream therefrom is markedly different for the two paths, in severe contrast to the usual assumption of parabolic flows of uniform streamwise pressure gradient over the cross-stream planes.

The predicted behaviour of the wall film cooling effectiveness is shown in figure C4 along the $z=0$ and $z=s/2$ lines. Notice that here, unlike the predicted behaviour with the parabolic procedure the off-centre line behaviour is generally similar to that predicted by the point source model.

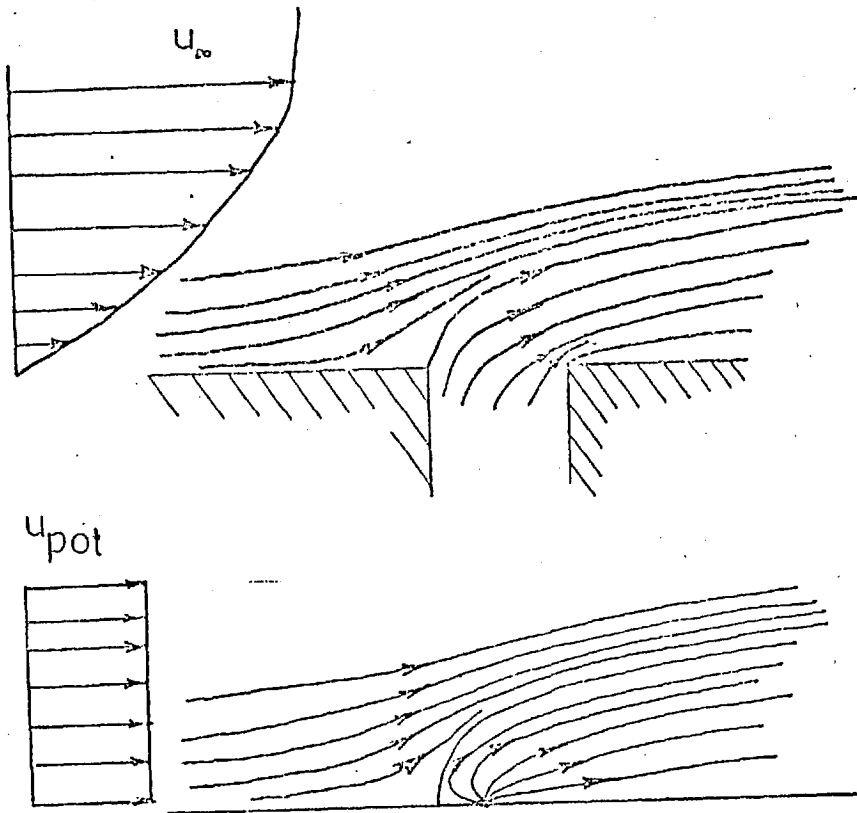


Fig. C1 : Streamline pattern.

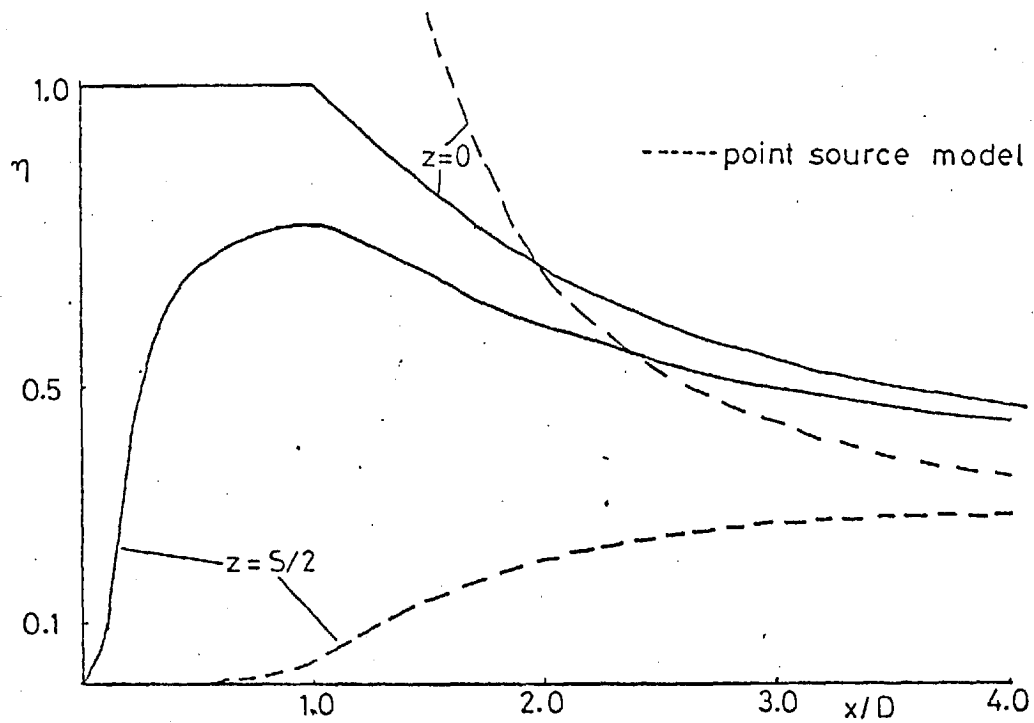


Fig. C2 : Predicted wall film cooling effectiveness

laminar flow, $\alpha=90^\circ$, $M=0.1$, parabolic

procedure.

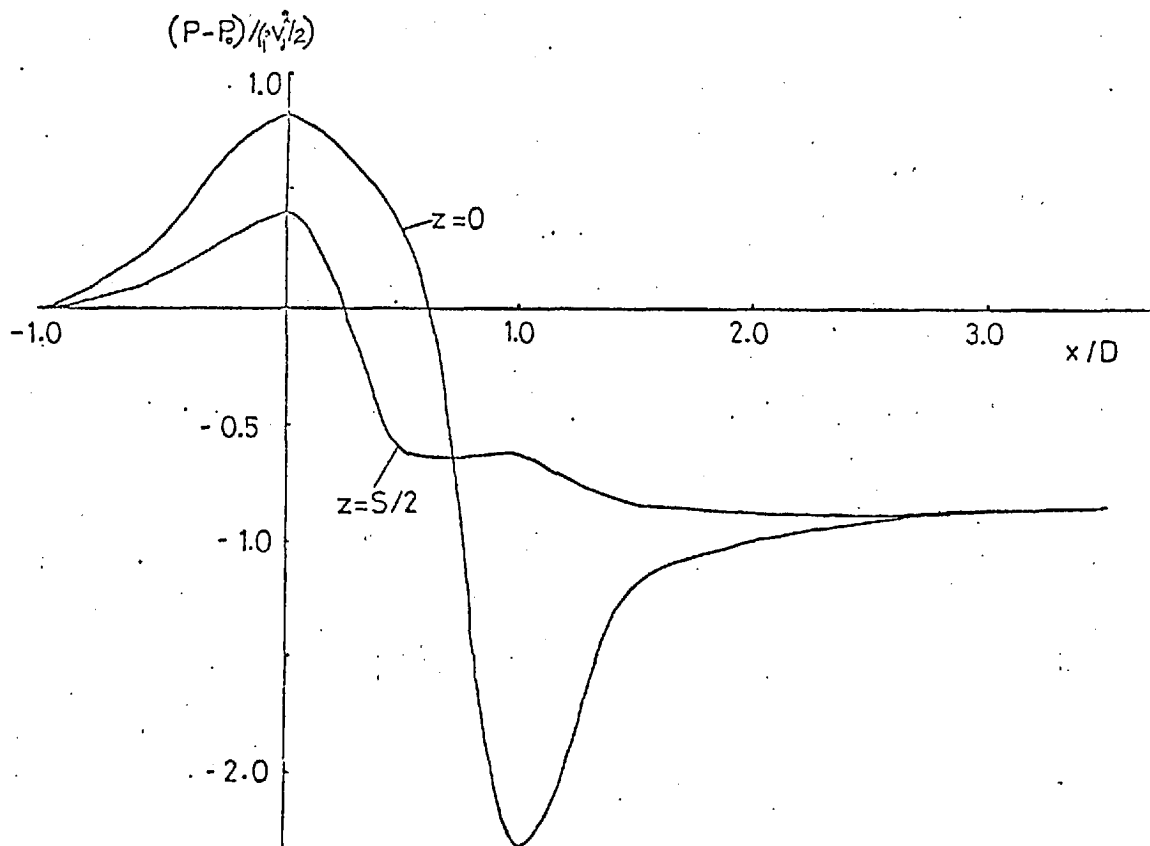


Fig. C3 : Predicted ~~surface~~ surface pressure distribution, laminar flow, $\alpha=90^\circ$, $M=0.1$.

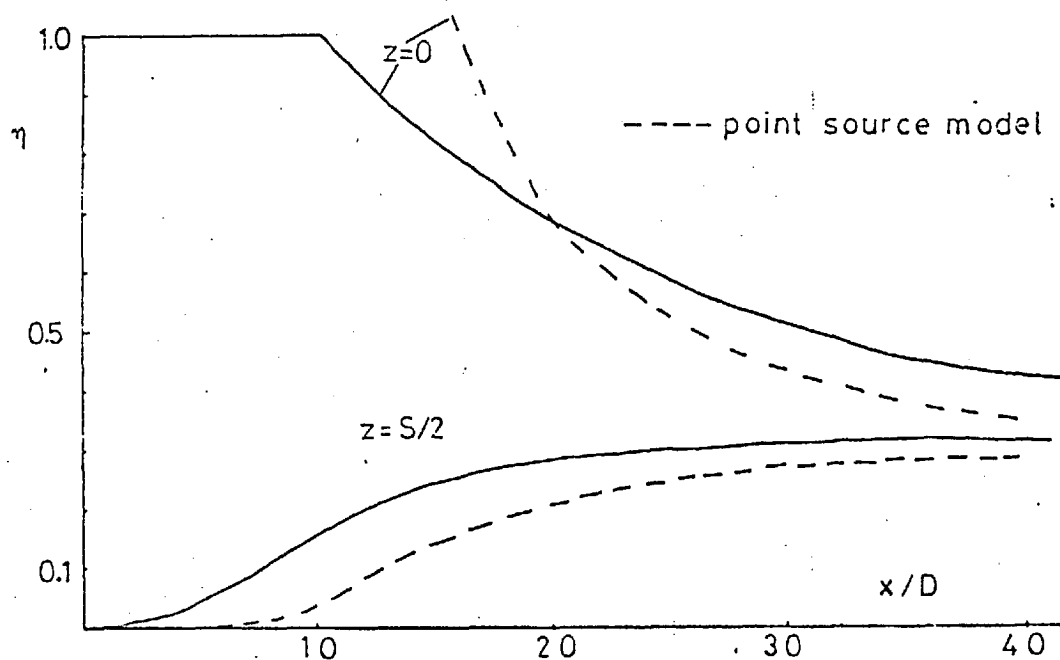


Fig. C4 : Predicted wall film cooling effectiveness, laminar flow, $\alpha=90^\circ$, $M=0.1$, semi-elliptic procedure.

75-WA/HT-109



\$3.00 PER COPY
\$1.50 TO ASME MEMBERS

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 printed only if the paper is published in an ASME journal or Proceedings. Released for general publication upon presentation. Full credit should be given to ASME, the Technical Division, and the author(s).

The Prediction of Three-Dimensional Discrete-Hole Cooling Processes: I-Laminar Flow

G. BERGELES

A. D. GOSMAN

B. E. LAUNDER

Mechanical Engineering Department,
Imperial College,
London, England

This paper describes the application of a three-dimensional finite-difference procedure to the problem of predicting the flow and thermal fields arising from the injection of fluid in discrete jets through a wall past which an external stream is flowing. The numerical scheme is of the "partially-parabolic" type originated by D. B. Spalding; this entails that, while the pressure field is held in three-dimensional arrays, the velocity field is stored only two-dimensionally, the velocities of the whole field being obtained by marching in the streamwise direction. This approach is substantially more economical of computer core than a fully three-dimensional treatment. Earlier experiences with using a strictly three-dimensional boundary-layer procedure (in which no variable is stored three dimensionally and computation proceeds without iteration in the direction of flow) had shown this approach to be incapable of handling the flow in the vicinity of the discharge hole, due to neglect of upstream transmission of pressure effects. Predictions of mean velocity and temperature for laminar flow are provided for the cases of a single row of holes inclined at 90, 45 and 35 deg to the plate surface and for a surface with multiple rows of holes in a staggered array. The above are for a uniform density flow and a uniform velocity external stream. A final example is presented, simulating typical operating conditions for a gas turbine blade cooled with a single row of holes aligned at 30 deg to the blade surface. Strong streamwise accelerations and density gradients are present. The results show, what has been observed experimentally, that due to the strong acceleration the lateral rate of spread is diminished. Moreover a counter-rotating vortex pair is created downstream from the hole which shifts the minimum effectiveness away from the midplane between the holes.

Contributed by the Heat Transfer Division of The American Society of Mechanical Engineers for presentation at the Winter Annual Meeting, Houston, Texas, November 30-December 4, 1975. Manuscript received at ASME Headquarters August 18, 1975.

Copies will be available until August 1, 1976.

Discussion on this paper will be received at ASME Headquarters until January 5, 1976.

THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS, UNITED ENGINEERING CENTER, 345 EAST 47th STREET, NEW YORK, N.Y. 10017

The Prediction of Three-Dimensional Discrete-Hole Cooling Processes: I-Laminar Flow

G. BERGELES

A. D. GOSMAN

B. E. LAUNDER

ABSTRACT

The paper describes the application of a 3-dimensional finite-difference procedure to the problem of predicting the flow and thermal fields arising from the injection of fluid in discrete jets through a wall past which an external stream is flowing. The numerical scheme is of the "partially-parabolic" type originated by D.B. Spalding: this entails that, while the pressure field is held in three-dimensional arrays, the velocity field is stored only two-dimensionally, the velocities of the whole field being obtained by marching in the streamwise direction. This approach is substantially more economical of computer core than a fully 3-dimensional treatment. Earlier experiences with using a strictly 3-dimensional boundary-layer procedure (in which no variable is stored three dimensionally and computation proceeds without iteration in the direction of flow) had shown this approach to be incapable of handling the flow in the vicinity of the discharge hole, due to neglect of upstream transmission of pressure effects.

Predictions of mean velocity and temperature for laminar flow are provided for the cases of a single row of holes inclined at 90°, 45° and 35° to the plate surface and for a surface with multiple rows of holes in a staggered array. The above are for a uniform density flow and a uniform velocity external stream. A final example is presented, simulating typical operating conditions for a gas turbine blade cooled with a single row of holes aligned at 30° to the blade surface. Strong streamwise accelerations and density gradients are present. The results show, what has been observed experimentally, that due to the strong acceleration the lateral rate of spread is diminished. Moreover a counter-rotating vortex pair is created downstream from the hole which shifts the minimum effectiveness away from the midplane between the holes.

NOMENCLATURE

Symbols

C_p	pressure coefficient, $(p - p_0)/\frac{1}{2}\rho_j u_{\infty}^2$
c_p	specific heat at constant pressure
D	hole diameter
J	diffusion flux of heat
M	injection ratio $(\rho_j V_j)/(\rho_{\infty} u_{\infty})$
p	static pressure
p_0	reference static pressure far upstream of injection
P_x	streamwise pitch
P_z	lateral pitch
T	temperature
$\bar{T}$	normalised temperature, $(T - T_{\infty})/(T_j - T_{\infty})$
u	streamwise velocity
v	vertical velocity
w	lateral velocity
u_e	streamwise velocity at exit of the hole
v_e	vertical velocity at exit of the hole

V_j	resultant coolant velocity at exit of the hole
u_{∞}	free stream velocity
x	streamwise direction
y	vertical direction
z	lateral direction

Greek Symbols

α	angle of injection
δ	boundary layer thickness
η	film-cooling effectiveness, $(T_w - T_{\infty})/(T_j - T_{\infty})$
λ	thermal diffusivity of coolant
μ	dynamic viscosity of the mixed stream
ν	kinematic viscosity of the mixed stream
ρ	density of the mixed stream
τ_{ij}	shear stress

Subscripts

w	wall value
j	conditions at hole exit
∞	undisturbed free-stream value

1. INTRODUCTION

In recent years there has been growing interest in the behaviour of jets of fluid discharged obliquely into a mainstream flow. The basic flow configuration is to be found in such practical problems as the dispersion of effluents from chimney stacks, the discharge of sewage (or hot water from power stations) into lakes or rivers, the injection of secondary air into combustion chambers and the cooling of leading stages of gas turbine rotors and stators. In the first two examples, the desired outcome is the dispersion of the jet fluid as rapidly and completely as possible into the main stream. The last example - that of turbine blade cooling - is the one that has motivated the present study, however. Here the intention is to arrange for jets, which emerge through holes in the blade surface, to form a "blanket" of cool fluid between the hot outer stream and the blade. These considerations make it desirable that the jet velocity should be but a small fraction of the mainstream velocity and that, where technically possible, the jet axis should be pointed downstream at an acute angle with the surface. In a large proportion of well-designed discrete-hole cooling systems the jet will, therefore, be strongly deflected immediately on discharge and either remain in contact with the surface or reattach so quickly that there will be negligible streamwise recirculation.

The flow field generated by discrete-hole injection is a three-dimensional one but, because of the special character of the flow noted above, numerical prediction schemes can make some simplification of the complete, three-dimensional elliptic equations of motion. Perhaps the obvious approach is to invoke the boundary-layer assumptions, in which the streamwise diffusion terms are discarded and the streamwise pressure gradient in any plane normal to the main flow is taken as uniform and known. When this is done, the equations become

parabolic in respect of the main flow direction, and velocities and pressures on any cross-stream plane become formally uninfluenced by those on parallel planes downstream. Thus, to obtain numerical solutions of this type of 3-dimensional flow it is possible to use a forward-marching procedure, in which velocities are calculated and stored at each plane in 2-dimensional arrays which are then successively overwritten as the computations proceed downstream. There now exist several finite-difference procedures of this kind, the most general of which is the scheme of Patankar and Spalding [1].

Earlier work by the present authors [2] using the procedure of [1] has however shown that the flow in the vicinity of the discharge holes could not be correctly resolved by a conventional boundary-layer treatment. The problem lies mainly in the behaviour of the pressure field, and to a lesser extent in the velocity field in the wake of the hole. Thus, even where the injection velocity is only 10% of the external-stream velocity, the jet nevertheless causes important modifications to the static pressure *upstream* of the hole, and provokes streamwise recirculation immediately downstream. The latter feature can, by approximate means, be accommodated by a boundary layer procedure, provided that the recirculation is small in extent and influence. The upstream pressure effects, however, introduce an element of ellipticity which precludes the use of a forward-marching procedure, and necessitates the storage of the complete (3-dimensional) pressure field.

Fortunately Pratap and Spalding [3] have recently devised an algorithm capable of solving problems of this kind - which they term "partially-parabolic" - and have applied it to flows in strongly curved ducts [4]. The new procedure, though obviously requiring more computer time and storage than a conventional boundary-layer treatment, achieves significant savings both in computing time and in the amount of core storage required over that of a discretisation of the complete three-dimensional equations of motion. Some further details will be given below.

The present contribution reports our work in obtaining numerical predictions of various discrete-hole-cooling configurations employing the partially-parabolic procedure. Section 2 below outlines the essential features of the analysis while Section 3 presents and discusses physical aspects of the solutions. The present computations relate entirely to laminar flow. Besides being both computationally and physically simpler than turbulent flow, laminar flow predictions have at least some practical relevance for flow over turbine blades because the very strong accelerations which occur over the blade prevent or delay the boundary layer becoming turbulent. The authors are currently engaged in obtaining predictions for turbulent flow which will appear in a later publication. The configurations studied in the present work are a single row of holes, where the hole axis makes angles of 90° , 45° and 35° with the surface and a multiple-row arrangement for 90° injection. Most of the computations relate to a flow of uniform density with a uniform-velocity external stream. For illustration, however, one run is presented where a strongly favorable pressure gradient exists and where fluid property variations representative of engine conditions are included.

2. METHOD OF ANALYSIS

2.1 Description of the Problem

The class of problems considered in the present investigation is depicted in Fig.2.1. The geometry is that of a flat plate, drilled with holes of diameter D whose axes make an angle α of between 0 and 90 degrees with the surface and are arranged in a single row or a staggered array, as shown in the diagram. The flow is laminar, and the pressure far from the surface is prescribed. Fluid is injected through the holes with uniform velocity V_j and temperature T_j . To be calculated are the distributions of velocity, pressure and temperature within the fields of influence of injection. The wall is assumed adiabatic and frictional heating neglected; of particular interest therefore is the distribution of film-cooling effectiveness, η , downstream from the holes.

2

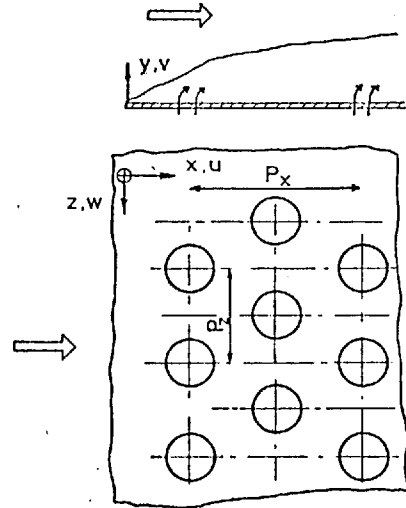


Fig. 2.1(a) The geometry

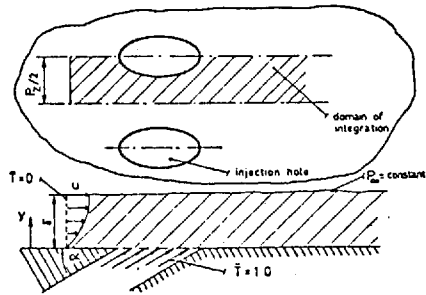


Fig. 2.1(b) Boundary conditions and domain of integration

2.2 Mathematical Description

The Cartesian directions x , y and z are identified (Fig.2.1) with the streamwise, vertical and spanwise directions respectively, and the corresponding velocity components are designated u , v and w ; the equations of motion and enthalpy within the framework of partially-parabolic flows, take the form:

Direction-x momentum

$$\frac{\partial}{\partial x}(\rho uu) + \frac{\partial}{\partial y}(\rho uv) + \frac{\partial}{\partial z}(\rho uw) = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial y}(\mu \frac{\partial u}{\partial y}) + \frac{\partial}{\partial z}(\mu \frac{\partial u}{\partial z}) \quad (2.1)$$

Direction-y momentum

$$\frac{\partial}{\partial x}(\rho uv) + \frac{\partial}{\partial y}(\rho vv) + \frac{\partial}{\partial z}(\rho vw) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} \quad (2.2)$$

Direction-z momentum

$$\frac{\partial}{\partial x}(\rho uw) + \frac{\partial}{\partial y}(\rho vw) + \frac{\partial}{\partial z}(\rho ww) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{zy}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \quad (2.3)$$

Continuity

$$\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0 \quad (2.4)$$

Enthalpy

$$\frac{\partial}{\partial x}(\rho u c_p T) + \frac{\partial}{\partial y}(\rho v c_p T) + \frac{\partial}{\partial z}(\rho w c_p T) = -\frac{\partial}{\partial y}(J_y) - \frac{\partial}{\partial z}(J_z) \quad (2.5)$$

where in tensor notation

$$\tau_{ij} \equiv \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (2.6)$$

$$J_i \equiv -\lambda \frac{\partial T}{\partial x_i} \quad (2.7)$$

and ρ , μ and λ are the density, viscosity and thermal conductivity, respectively. Note the absence of streamwise diffusion terms and the requirement that (strictly) u must be everywhere positive.

Solutions of the foregoing equations would typically be sought over the region shown cross-hatched in Fig.2.1(b) which depicts a row of inclined holes. The lateral boundaries are located at planes of symmetry, and the upstream and outer-edge boundaries are located in regions of undisturbed flow. In the present examples, the upstream boundary is located 1.5 diameters upstream of injection, the flow being prescribed as two dimensional. The boundary-layer velocity profile is given by:

$$\frac{u}{u_\infty} = \frac{3}{2} \left(\frac{y}{\delta} \right) - \frac{1}{2} \left(\frac{y}{\delta} \right)^3 \quad (2.8)$$

where u_∞ and δ are the free-stream velocity and boundary-layer thickness, respectively. Equation (2.8) is known to fit closely the exact solution to the laminar boundary in zero pressure gradient. The velocity components u_e and v_e of the fluid existing from the hole are calculated from:

$$\left. \begin{aligned} u_e &= V_j \cos \alpha \\ v_e &= V_j \sin \alpha \end{aligned} \right\} \quad (2.9)$$

where V_j is the injection velocity. The dimensionless temperature T is set equal to unity in the injected fluid and zero in the free stream.

2.3 Numerical Solution Procedure

The main elements of the partially-parabolic finite-difference procedure (of which full details are given in [3]) may be summarized as follows:

1. The dependent variables u , v , w , p and T are computed on grids superimposed on the domain of solution, and arranged such that, while pressure and concentration share the same locations, the velocity components lie mid-way between the pressures which drive them. The grid is nominally Cartesian, but may expand or contract in the vertical direction to conform with the boundary-layer profile.
2. Finite-difference equations (fde's) are derived for u , v , w and T : these relate each variable at a point to its nearest neighbours in the same cross-stream plane and to the adjacent upstream (but not downstream) neighbour. In the case of the velocity equations, the neighbouring pressures in the direction of the component in question appear as well.
3. Pressure is obtained from its own equation, derived by combining the fde's for momentum and continuity. Unlike the other members of the set, this equation relates to the nearest grid neighbours in all directions, including downstream.
4. Starting from initial guesses for the velocity, pressure and temperature fields, the procedure operates by scanning the cross-stream planes one by one, beginning from the upstream end and solving at each plane for all variables, on the assumption that the values on neighbouring planes are known. In the case of the velocities, which do not depend on values at downstream planes, the latter supposition

Boundary	u	v	w	p	T
bottom (plate) face	0	0	0	-	$\frac{\partial T}{\partial y} = 0$
at hole exit	u_e	v_e	0	-	T_j
top (free stream) face	$u = u_\infty(x)$	-	0	$p_\infty(x)$	T_∞
upstream face	$u = u(y/\delta)$	$v = v(y/\delta)$	0	-	T_∞
downstream face	-	-	-	extra-polation	-
edges	$\frac{\partial u}{\partial z} = 0$	$\frac{\partial v}{\partial z} = 0$	0	$\frac{\partial p}{\partial z} = 0$	$\frac{\partial T}{\partial z} = 0$

Table 2.1 Summary of boundary conditions

is a reasonable one (likewise for temperature). Thus provided that the pressures were correctly specified, one scan of the field would secure the solution to the problem (this of course is precisely the way conventional boundary-layer methods operate). In practice, the effects of injection will necessitate that adjustments be made to the pressures and hence the velocities: thus, the scanning procedure is repeated in an iterative fashion using the pressure field of the previous iteration until a satisfactory solution is achieved.

5. The procedure therefore preserves as far as possible the limited-storage requirements of boundary-layer methods, requiring only that the static pressure, whose influence is transmitted in all directions, be stored 3-dimensionally.

A summary of the boundary conditions applied is provided in Table 2.1. The results which are presented in the next section were obtained, after grid-refinement tests, with grids which typically comprised 70, 18 and 17 planes in the x , y and z directions respectively, spaced so as to be concentrated in regions of steep gradients. Computing times per run and storage requirements on a CDC 6600 machine were about 14 min and 50 K respectively.

3. RESULTS

3.1 Single row of holes, 90° injection

For this series of calculations normal injection was simulated through a single row of holes with pitch P equal to $2D$. The upstream boundary was located $1.5D$ from the leading edge of the hole, and the initial conditions were specified as $\delta = 5.5D$ and $R_\delta (\equiv u_\infty \delta / \nu) = 690$. Results will be presented for values of the injection ratio $M (\equiv \rho_j V_j / \rho_\infty u_\infty)$ of 0.1 and 0.2. For these and subsequent examples the Prandtl number was 0.70.

An immediate impression of the importance of elliptic pressure effects may be gained from Fig.3.1(a), which shows for $M = 0.2$ surface contours of the pressure coefficient C_p , the reference pressure is that in the undisturbed flow far away from the holes. (It may be remarked that the choice of $\rho_j V_j u_\infty$ as the normalising quantity was made on the basis of experiments in turbulent flow [5] where this practice was found nearly to collapse the data for the two injection ratios onto identical curves.) Evidently there is an appreciable upstream effect even for the smaller of the injection ratios: the 10% contour extends to about one diameter. Also evident are appreciable lateral and downstream variations, with the latter persisting for longer distances than upstream, and revealing the presence of adverse pressure gradients in the region immediately downstream of the hole. Over the hole itself there exist strong favorable pressure gradients.

The predicted flow pattern for $M = 0.2$ is conveyed by Figs.3.1 (b), (c) and (d). The first of these shows the velocity vectors in the

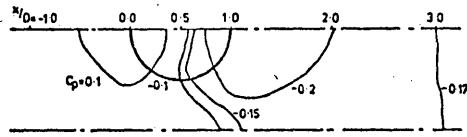


Fig.3.1(a) Isobars for $M = 0.2$

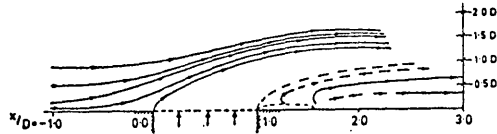


Fig.3.1(b) Velocity directions in the symmetry plane

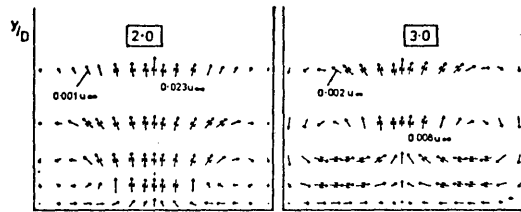


Fig.3.1(c) Velocity vector field in cross-stream plane

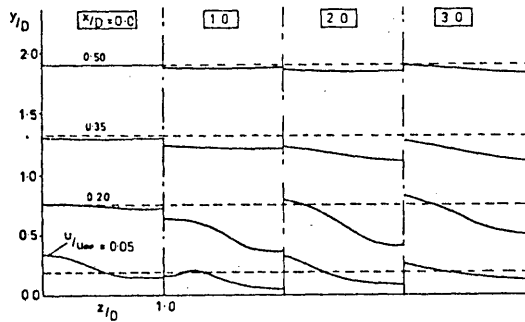


Fig.3.1(d) Contours of u velocity in cross-stream planes

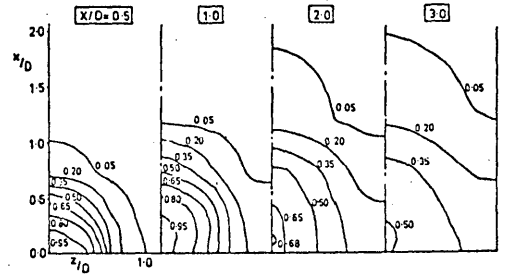


Fig.3.1(e) Normalised isotherms in cross-stream planes

symmetry plane which bisects the hole: the action of the pressure field in deflecting upwards the oncoming stream can be clearly seen (it is worth recalling that a conventional boundary-layer procedure is incapable of simulating this effect), as can the further deflection over the hole provoked by the injected fluid. Immediately downstream is a small zone of backwards flow which, although strictly not allowed by the procedure, can be patched over by replacing the calculated small negative values by zero; this practice does not appreciably affect the results elsewhere. Fig.3.1(c) shows projections of the velocity vectors into cross-stream planes at various locations, starting from the upstream edge of the hole ($x/D = 0.0$) and proceeding downstream. The symbol z marked across a vector line denotes that it has been drawn at less than full scale to avoid cross-over of the lines. The first two plots merely serve to confirm the statements made earlier about the initial upward deflection of the flow: the remaining ones however reveal a new, but expected, feature; namely, the formation of the familiar streamwise vortex pair in the wake of the hole, which is a manifestation of the vorticity added to the flow by the fluid injection. The remaining set of velocity plots in Fig.3.1(d) show contours of the streamwise component at various cross-stream planes, in which the dashed lines represent the contours which would prevail in the absence of injection. The contours confirm the expected retardation of the flow at the leading edge, the acceleration over the hole itself, and the deceleration in the downstream wake. Also shown is the tendency for the lateral variations to diminish with distance from the holes. The general hydrodynamic behaviour for $M = 0.1$, although not shown, is qualitatively similar, but the disturbances produced by the jet are of course less pronounced.

Figure 3.1(e) displays contours of T in cross-stream planes for $M = 0.2$. These show how the thermal field spreads initially in an almost radially symmetrical fashion and is subsequently distorted by the velocity field and the interaction with neighbouring holes. Of particular interest are the centre panels, which illustrate the action of the streamwise vortices in bringing fluid of low normalised temperature in from the sides, thereby causing "pinching" of the contours near the plate, and the displacement of the maximum value of T from the surface.

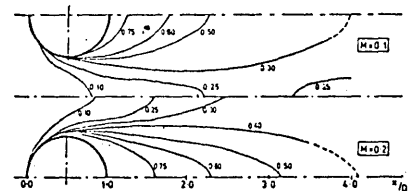


Fig.3.2 Wall film-cooling effectiveness for $M = 0.1$ and $M = 0.2$

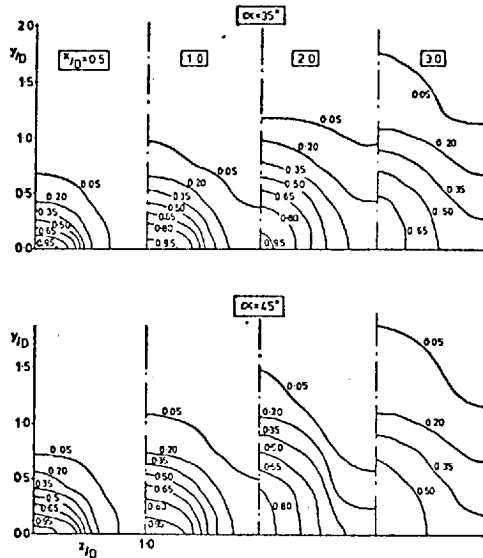


Fig.3.3 Normalised isotherms in cross-stream planes $\alpha = 35^\circ$, $\alpha = 45^\circ$.

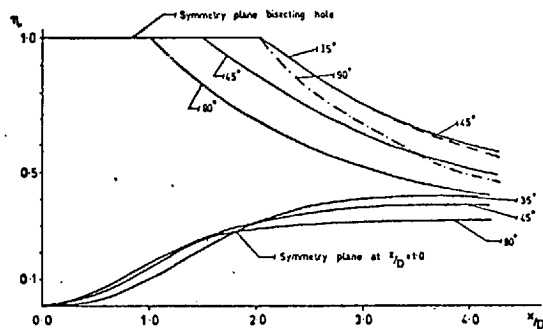


Fig.3.4 Effect of injection angle on surface effectiveness along symmetry lines. Single row, $P_z = 2.0$, $M = 0.2$

Figure 3.2 shows the wall film-cooling effectiveness for $M = 0.1$ and 0.2 . Here we note the rather rapid lateral spread around the hole which seems to be a consequence of the strong lateral motions induced by the jet, and the comparatively low velocity of the fluid layers fed by these motions. The effect of increasing M is to raise the effectiveness. This behaviour is to be expected at the low values of M considered in the present study where the jet fluid reattaches to the surface a short distance behind the hole.

3.2 Single Row of Holes, Various Injection Angles, $M = 0.2$

Figure 3.3 shows the effect on the temperature field of varying the angle of injection, with all other conditions held constant and $M = 0.2$. The upper sequence of panels contains results for $\alpha = 35^\circ$, and the lower set for $\alpha = 45^\circ$. Comparisons of these predictions, and those for $\alpha = 90^\circ$ in Fig.3.1(c) shows that as the angle is reduced, the rate of vertical spread is diminished.

An impression of the effect of injection angle on surface effectiveness may be seen in Fig.3.4, which displays the variation along lines passing through the hole axis and mid-way between adjacent holes, for $\alpha = 35^\circ$, 45° and 90° . It is seen that reducing the angle of injection from 90° to 45° reduces the streamwise rate of decay of η

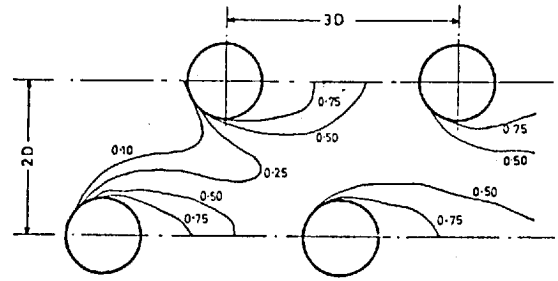


Fig.3.5(a) Film-cooling effectiveness for multi-row hole geometry

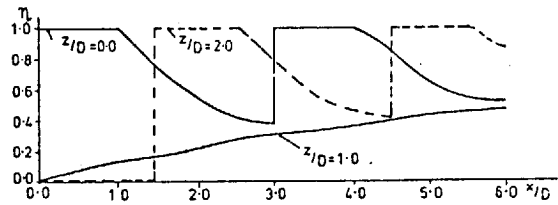


Fig.3.5(b) Film-cooling effectiveness along and mid-way between symmetry planes

(qualitatively in agreement with experimental data of turbulent flow [6]) but that reducing the angle further to 35° has relatively little effect. (The broken lines show the rate of decay along the $z = 0$ line; there has been a shift of origin of the x axis to make the downstream end of the hole occur at the same position for all three angles of injection.) It is seen that immediately behind the hole, the effectiveness along $z/D = 1.0$ decreases as injection angle is reduced. This is attributable to changes in the flow field: the smaller the injection angle the less the disturbance created by the jet; in particular the static pressure variation around the hole is reduced. As a result w velocities are weaker and the displacement thickness of the wake is considerably less. Further downstream, however, concentration levels become significantly higher along $z/D = 1$ the smaller the injection angle implying a faster lateral spreading rate. This behaviour is qualitatively different from that observed in turbulent flow experiments. There is not necessarily any conflict in these results, however. In turbulent flow the effective transport coefficients are large wherever the mean rates of strain are large (as indicated, for example, by the mixing-length hypothesis). It is thus quite possible that in a turbulent flow the reduction in strain rates that occurs as the angle of injection is progressively decreased reduces the transport coefficients sufficiently for the lateral rate of spread to fall. It is noted that the present results all show a tendency to approach two-dimensional behaviour at about the same rate.

3.3 Multi-row Injection: 90° Holes

When only a single row of cooling holes is used most blade-cooling applications will require higher injection rates than are considered in the present paper. Raising the level of M , however, tends to make the jet lift away from the surface promoting increased mixing (and hence hastening the rate of dilution of the coolant) and adversely affecting the aerodynamic characteristics of the blade. For this reason attention is being given to cooling systems employing more than one row of holes: by this means sufficient coolant can be introduced to the blade surface while keeping injection velocities at acceptably low levels. Figures 3.5 and 3.6 provide illustrations of the calculated behaviour of a multi-row injection system for $M = 0.1$. As before, the boundary-layer thickness is taken as 5.5δ but now the lateral pitch between holes is 4.0 with successive rows being staggered and repeated rows occurring every 3 diameters downstream. The results shown in these figures have been obtained by making a number of computations in

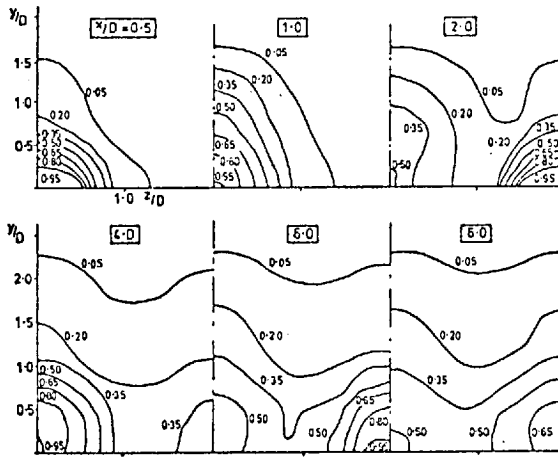


Fig.3.6 Contours of normalised isotherms in cross-stream planes for multi-row geometry

which the finite-difference grid covered only a portion of the total flow domain, the grid being successively moved downstream until the region shown in Fig.3.5 had been covered.

Surface effectiveness contours are shown in plan view in Fig.3.5 (a) while the variation of effectiveness along the two lateral edges of the flow domain and along the line midway between the edges is shown in Fig.3.5(b). In addition, an impression of the temperature field in the boundary layer is provided by the normalised isotherms at various y - z planes shown in Fig.3.6. It may be seen that immediately behind the leading hole the normalised temperature falls off more rapidly than for the single row test displayed in Fig.3.2. This seems to be mainly due to the greater lateral spacing between holes in the same row. Overall, however, the multi-hole arrangement is

superior to the single-row configuration. For example, after the second row, the amount of injected fluid is the same as for the single row test with $M = 0.1$; the minimum concentration level at $x/D = 2.9$ (i.e. just upstream of the third row) is 0.29 in Fig.3.5 compared with 0.26 in Fig.3.2. Correspondingly, after the fourth row of holes the amount of coolant is the same as for the single row test with $M = 0.2$. Now, in the latter case, computations extended only three diameters behind the injection holes ($x/D = 4$). From Fig.3.4 it is seen that at that station the minimum concentration level of 31% is considerably less than the 44% at $(x/D) = 5.9$ for the multi-hole injection. There will also be aerodynamic advantages to the multi-hole geometry that we do not attempt to quantify in the present paper.

3.4 Single-row Injection with Density Gradient and Free-stream Acceleration

The final example relates to conditions approximating more closely, than the examples considered above, the kind of situation which might be encountered under actual operating conditions. The coolant is injected through a single row of 30° holes, the mass velocity ratio, M , being 0.26 and the ratio of lateral pitch to hole diameter being 2.5. The ratio of injectant to main-stream densities is approximately 1.4:1 and the streamwise velocity gradient du_∞/dx is virtually constant, the initial value of $(\nu/u_\infty^2) du_\infty/dx$ being equal to 0.5×10^{-6} . The viscosity is assumed to be uniform. In this example computations with the partially-parabolic procedure extended from 2 diameters upstream of the hole to 7.0 diameters downstream; the incoming boundary layer was now taken as 0.2 hole-diameters thick which again seemed close to probable values met in practice. The Pohlhausen 4th-order polynomial was used to prescribe the distribution of velocity in the boundary layer. The variation of free-stream velocity implied by the acceleration parameter was applied at the free stream edge of the flow domain. In practical computations, the finite-difference calculations would need to extend tens of hole diameters downstream rather than just a few. One could proceed in the same way as with the multi-row computations, covering a small strip of the flow at a time. This would be an unnecessarily expensive practice,

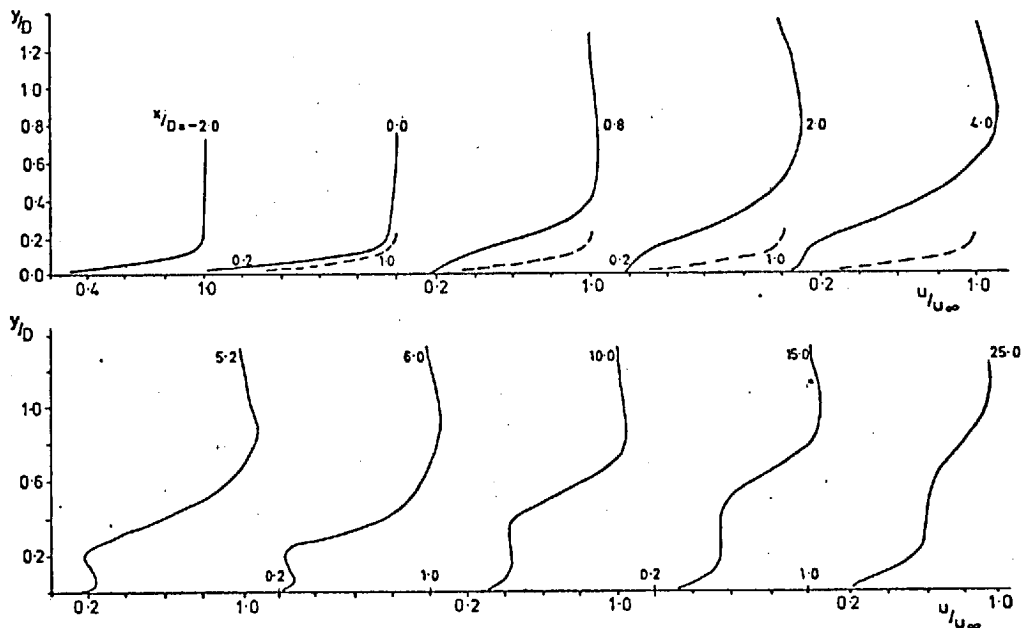


Fig.3.7(a) Streamwise velocity profiles in the symmetry plane through a hole. Single row, $\alpha = 30^\circ$, non-uniform density, free-stream acceleration.

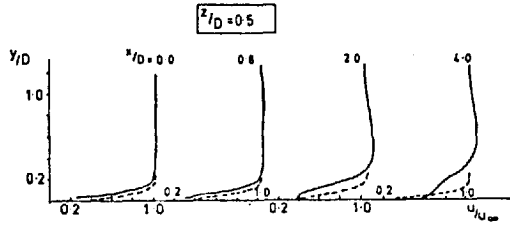


Fig. 3.7(b) Streamwise velocity profiles on planes parallel to the symmetry plane; $z/D = 0.5$

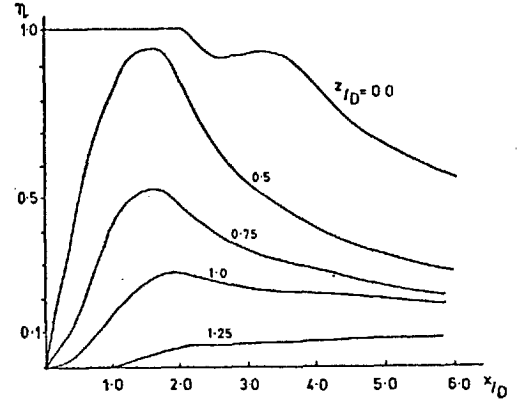
however, since the cross-stream pressure variations become very small a few hole diameters downstream. Instead we have employed the orthodox 3-dimensional parabolic treatment of Patankar and Spalding [1] beginning the computation 5 diameters downstream from the leading edge of the injection holes using, as initial profiles, the values given at that section by the partially-parabolic method. The computation proceeds downstream in a non-iterative marching fashion with the streamwise pressure gradient being treated as uniform at any section in the x -momentum equation.

The development of the streamwise velocity profile along a plane through the hole centre and for $z/D = 0.5$ is shown in Figs 3.7. For reference the mean velocity profile at the upstream station ($x/D = -2.0$) is shown by a broken line on all figures. The curves are normalised by the local free-stream velocity. Referring to Fig. 3.7(a), there is only a small displacement of the boundary layer upstream of the hole but a substantial displacement of the profile over the hole itself. Note that the profile at $x/D = 0.8$ displays a slip velocity at $y/D = 0$ due to the streamwise component of velocity of the injected stream. Behind the hole a large wake in the boundary layer is evident. The jet itself lifts away from the surface but has re-attached by $x/D = 5.2$ as evidenced by the small peak in velocity close to the wall. As the shear flow develops downstream the velocity maximum disappears leaving a patch of slow moving, virtually uniform-velocity fluid close to the wall. With further development downstream this patch accelerates more rapidly than the free stream fluid (as required by Bernoulli's equation) and a "new" boundary layer may be seen developing beneath it.

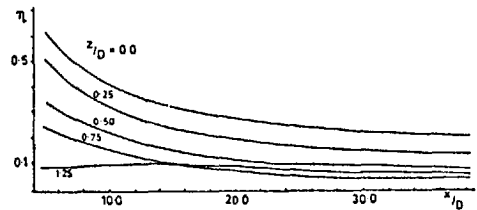
The off-centre line behaviour in the vicinity of the hole is shown in Fig. 3.7(b). For $z/D = 0.5$ the flow development upstream and around the hole appears similar to that along $z/D = 0$ discussed above. The profile at $x/D = 4.0$ is, however, strikingly different from the corresponding profile in Fig. 3.7(b). The flow near the wall has been strongly accelerated by the "blockage" effect of the injectant and consequently is moving faster than the initial boundary layer fluid. The effect dies off rapidly at greater distances from the hole centre line and at $z/D = 1.25$, which is the mid-plane between adjacent holes, the change in velocity from that of the initial profile is barely significant.

The variation of film-cooling effectiveness along a number of constant- z/D lines is shown in Figs 3.8(a) and (b), which relate to the partially-parabolic and the parabolic regions of computation respectively. In the former, the main points to note are the rather surprising dip along the axis immediately behind the hole and the steep rise in normalised temperature at points off the centre-line. The first effect is due to the jet lifting off the surface and then reattaching, discussed earlier. The second is caused mainly by the strong lateral motions around the jet (the static pressure being higher along $z/D = 0$ for $x/D < 1$). The downstream region shown in Fig. 3.8(b) displays a gradual decay of effectiveness with distance. An interesting

* The hole extends from $x/D = 0$ to $x/D = 2.0$ since the holes are aligned at 30° with the surface.



(a) The near-field partially parabolic region



(b) The far-field parabolic region

Fig. 3.8 Wall film-cooling effectiveness

feature is the fact that for values of x/D greater than about 12 the minimum effectiveness occurs not along the bisector of the injection holes but at $z/D \approx 0.75$. The cause of this behaviour may be inferred from Fig. 3.9 which shows computer-drawn velocity vectors in the y - z plane at $x/D = 32$. Injecting fluid through the holes creates two eddies rotating in the y - z plane (i.e. with axes in the x -direction). One of these, induced directly by the injection, rotates clockwise (viewed from the upstream side) and is located in the left-hand half of the section shown in Fig. 3.9. The second vortex may be thought of as being an indirect consequence of the jet; it is the so-called "horseshoe" vortex in which the vortex lines in the boundary layer upstream of the hole bend around the jet giving an extra component to the streamwise vorticity. This vortex rotates counter clockwise and is located in the right-hand half of Fig. 3.9. It is evident from this figure that the right-hand member of this counter-rotating vortex pair is very much weaker than the left-hand one. Nevertheless, together their action is to bring high temperature fluid down towards the surface displacing the cooler fluid to each side.* This pattern of development may clearly be seen in Fig. 3.10 which shows normalised temperature contours in the y - z plane at different stations. A further item to

* A referee has commented that his turbulent flow measurements exhibit the same qualitative behaviour.

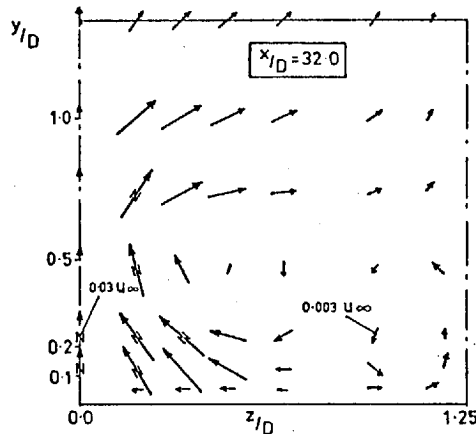


Fig.3.9 Velocity direction lines in $y-z$ plane

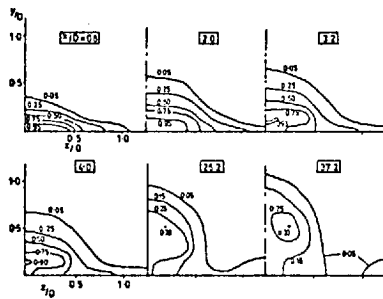


Fig.3.10 Normalised isotherms in cross-stream planes

be noted from this figure is that the peak normalised temperature at any section occurs at some distance away from the wall. This is due, at first to the fact that the jet lifts away from the surface immediately behind the slot; further downstream the phenomenon may be attributed to the counter-rotating vortices discussed above. It may be questioned why the secondary vortex should be present in this flow but not apparently in the other examples of single row injection discussed earlier (for example there is no sign of a secondary vortex in Fig.3.1(c)). The most probable explanation seems to be that, in the earlier examples, the upstream boundary layer was much thicker relative to the hole diameter than in the last case. Consequently most of the boundary-layer vorticity would have passed over the top of the injection hole without the vortex lines being significantly bent.

The flow conditions for this final test corresponded with those of some measurements undertaken by Rolls Royce (1971) Ltd. of flow over actual blades. In Fig.3.11 comparison is made between the span-wise averaged values of effectiveness measured in that study and the present predictions. It is seen that though the measured values are consistently a little higher than the calculated ones the general level of agreement may be regarded as satisfactory.

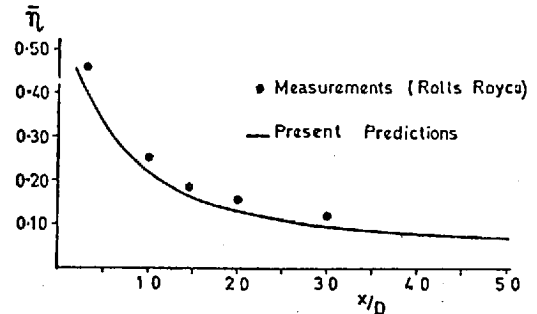


Fig.3.11 Span-wise averaged cooling effectiveness: comparison with experiment.

4. CONCLUDING REMARKS

The present study has successfully applied a 3-dimensional, finite-difference procedure to the computation of discrete-hole injection processes. Through the use of the partially-parabolic scheme [3] calculations of the complex flow structure around the injection holes have been made without the need for unattractively large computer core storage.

The present work is a first step towards providing a tested computational scheme for turbulent flows of this kind. We have been unable here to draw comparisons with experiment for no flow field data of laminar, discrete hole injection are known to us. This might be an area where a well-documented experimental investigation would be extremely valuable.

A number of interesting phenomena have been thrown to light by the computation, in particular it is suggested that the thickness of the boundary layer upstream of the hole has a strong influence on the character of the flow downstream. If this proves to be the case, it suggests that designers should be rather cautious in using geometric and velocity-ratio criteria alone in estimating the cooling effectiveness of a given configuration of holes.

5. ACKNOWLEDGEMENTS

We gratefully acknowledge the support of this work by the Ministry of Defence Procurement Executive and the Ministry's permission to publish the results. The computer program used in the calculations has been developed from the 3-dimensional parabolic code STABL loaned to us by Combustion, Heat and Mass Transfer Ltd. We also wish to express our thanks to Rolls Royce (1971) Ltd. for allowing us to make comparisons with their as yet unpublished measurements of flow over a blade cooled by discrete hole injection.

Several helpful discussions and exchanges with Mr. V.S. Pratap and Professor D.B. Spalding are acknowledged with thanks.

REFERENCES

- 1 Patankar, S.V. and Spalding, D.B., "A calculation procedure for heat, mass and momentum transfer in 3-dimensional parabolic flows," *International Journal of Heat and Mass Transfer* 15, 1972, pp.1787-1806.
- 2 Bergeles, G., Gosman, A.D. and Launder, B.E., "The prediction of discrete hole cooling: A progress report on experience with parabolic and partially-parabolic versions of STABL," Aug. 1974, Imperial College, Mechanical Engineering Department Report.

- 3 Pratap, V.S. and Spalding, D.B., "A calculation procedure for partially-parabolic flow situations," IHS/74/35, Dec. 1974, Imperial College, Mechanical Engineering Department Report.
- 4 Pratap, V.S. and Spalding, D.B., "Numerical computations of the flow in curved ducts," IHS/75/3, Feb. 1975, Imperial College, Mechanical Engineering Department Report.
- 5 Bergles, G., Gosman, A.D. and Launder, B.E., "The near-field character of a jet discharged through a wall at 90° to a main stream," ASME paper to be presented at Winter Annual Meeting, November 1975.
- 6 Ramsey, J.W. and Goldstein, R.J., "Interaction of a heated jet with a deflecting stream," NASA CR-72613, April 1970.

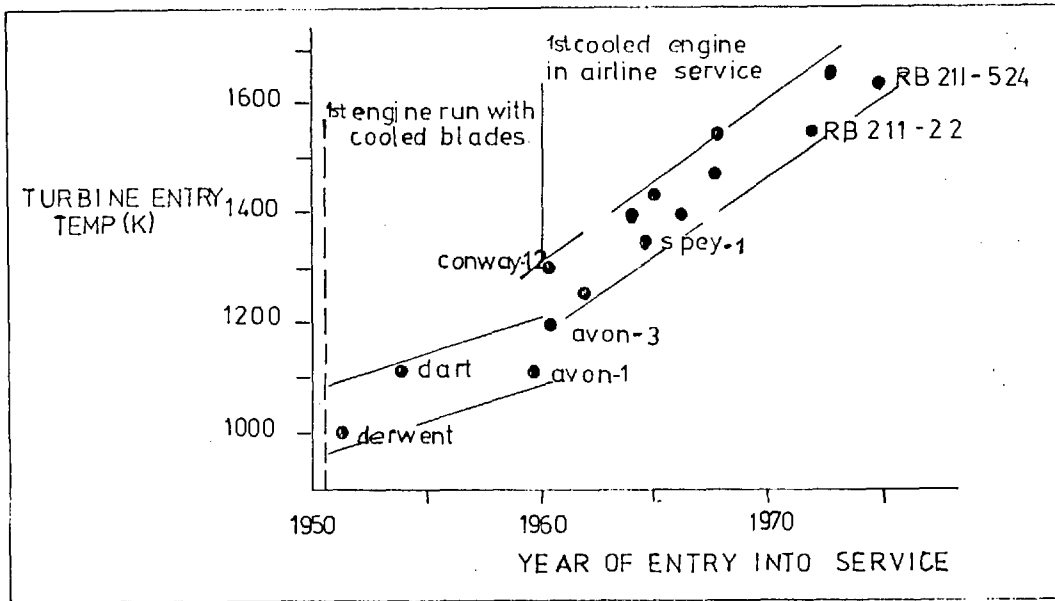


Fig. 1 : Turbine entry temperature (taken from (1))

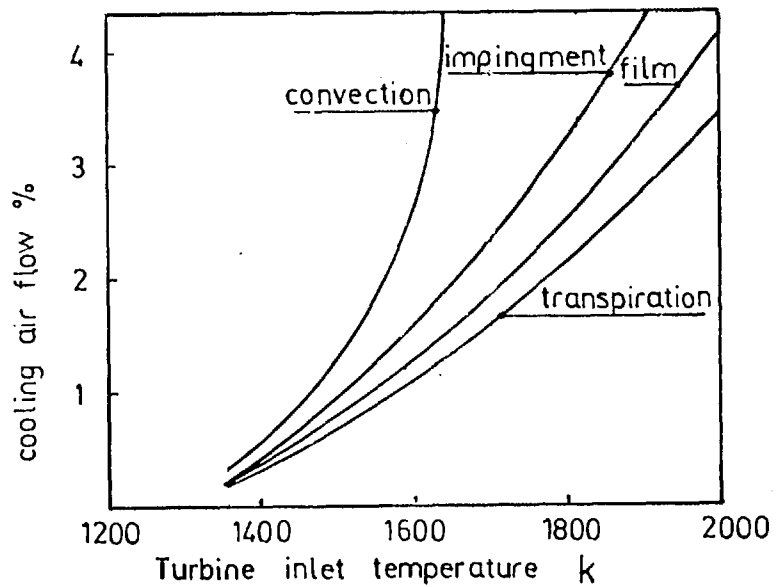


Fig. 2 : Comparison of cooling systems
(taken from (3))

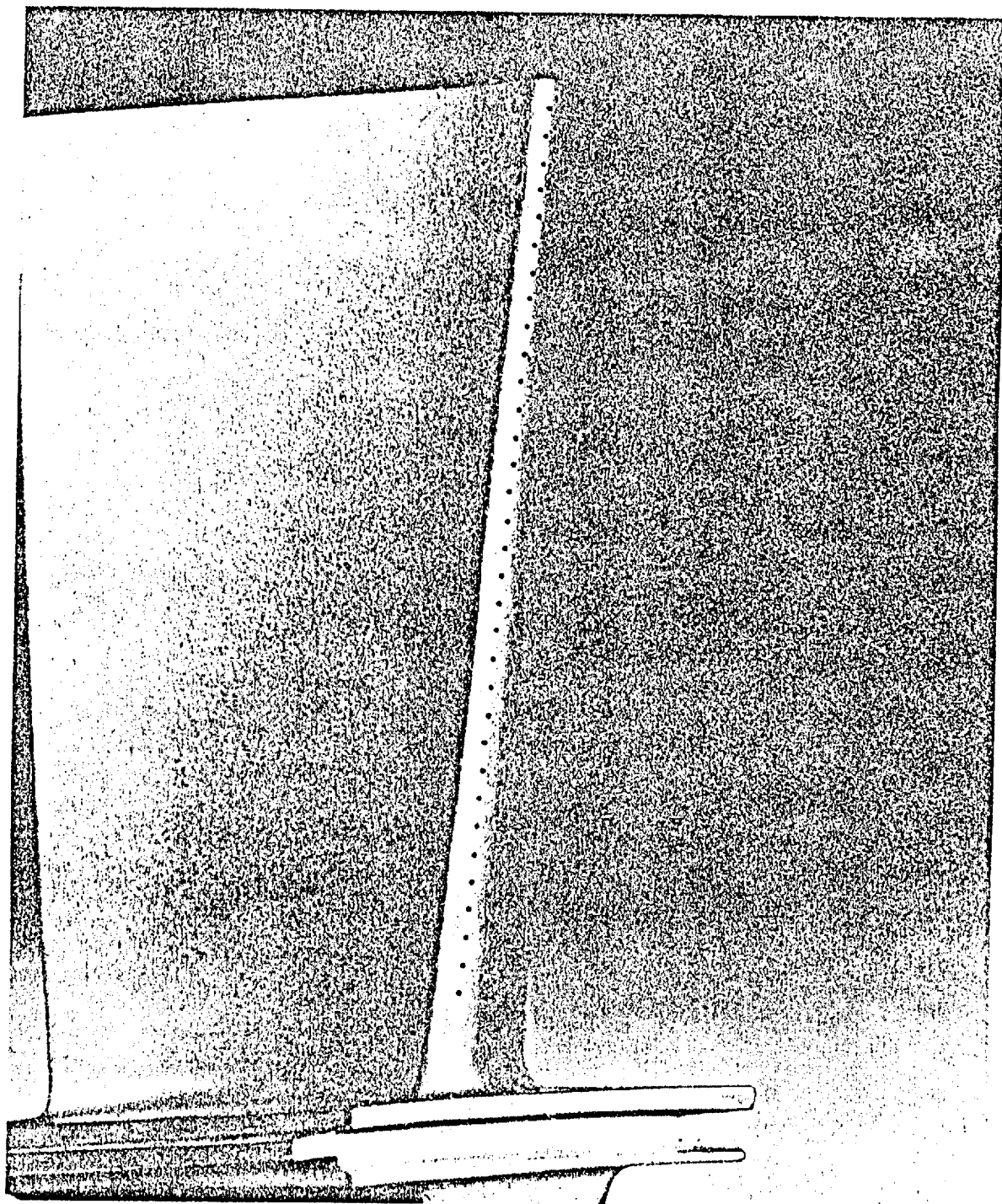


Fig. 3 : Experimental turbine blade (taken from (3))

ENGINE	THRUST	TET	
-22C	38750 LB	1489 °K	1060 °C 1085 °C
-22B	42000 LB	1493 °K	1035 °C 960 °C
-22B	42000 LB PARTIAL HP FEED	1493 °K	935 °C 950 °C
-524	50000 LB SPLIT HP FEED	1548 °K	935 °C 950 °C

TET. Turbine-entry temperature.

Fig. 4 : The RB211 turbine blade cooling system
(taken from (4))

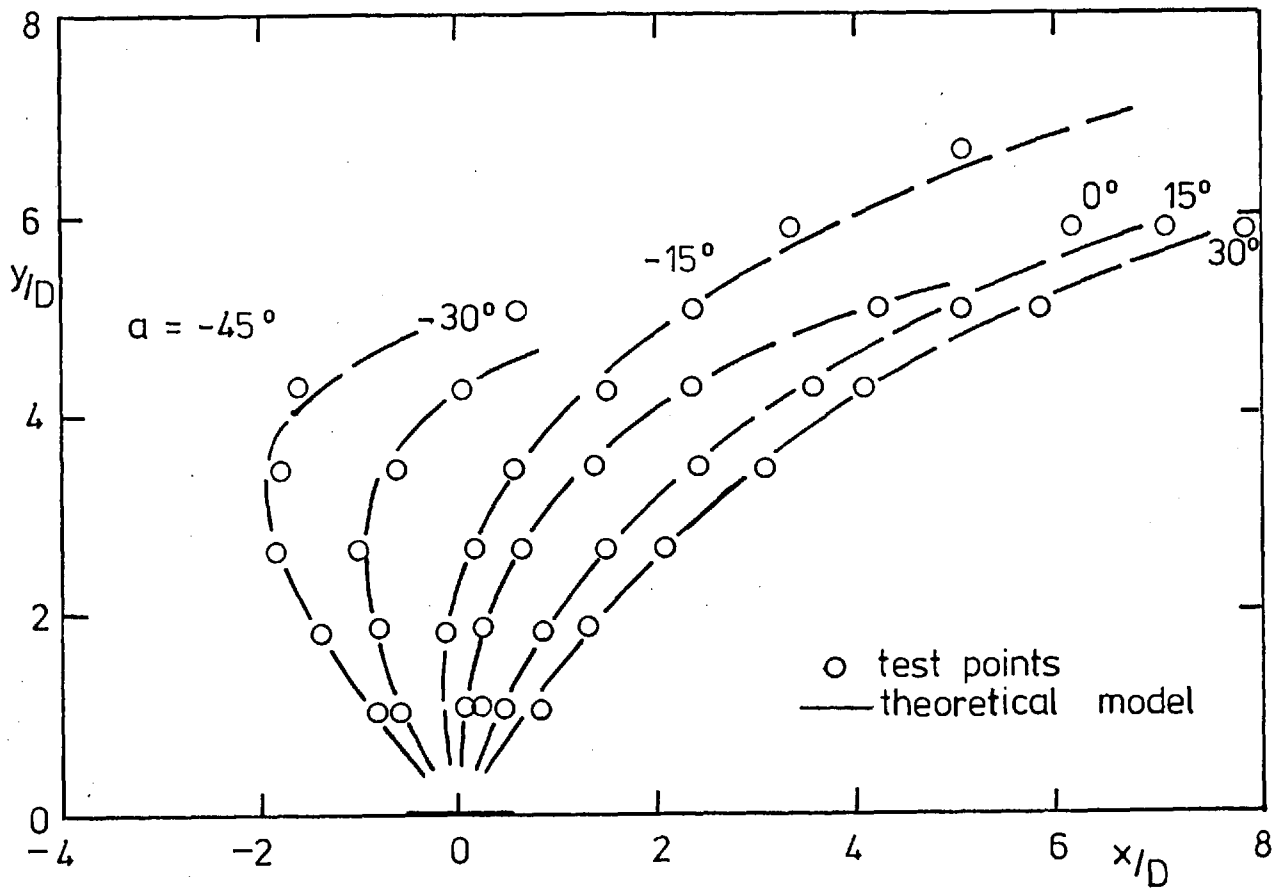


Fig. 5 : Predicted jet trajectory (taken from (5))

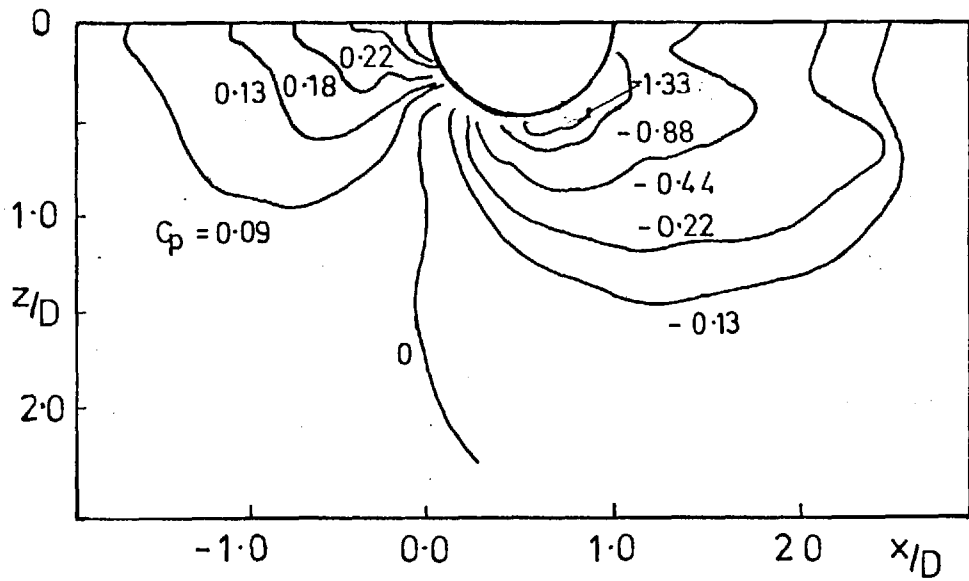


Fig. 6 : Static pressure distribution, $M=2.26$ (taken from (8))

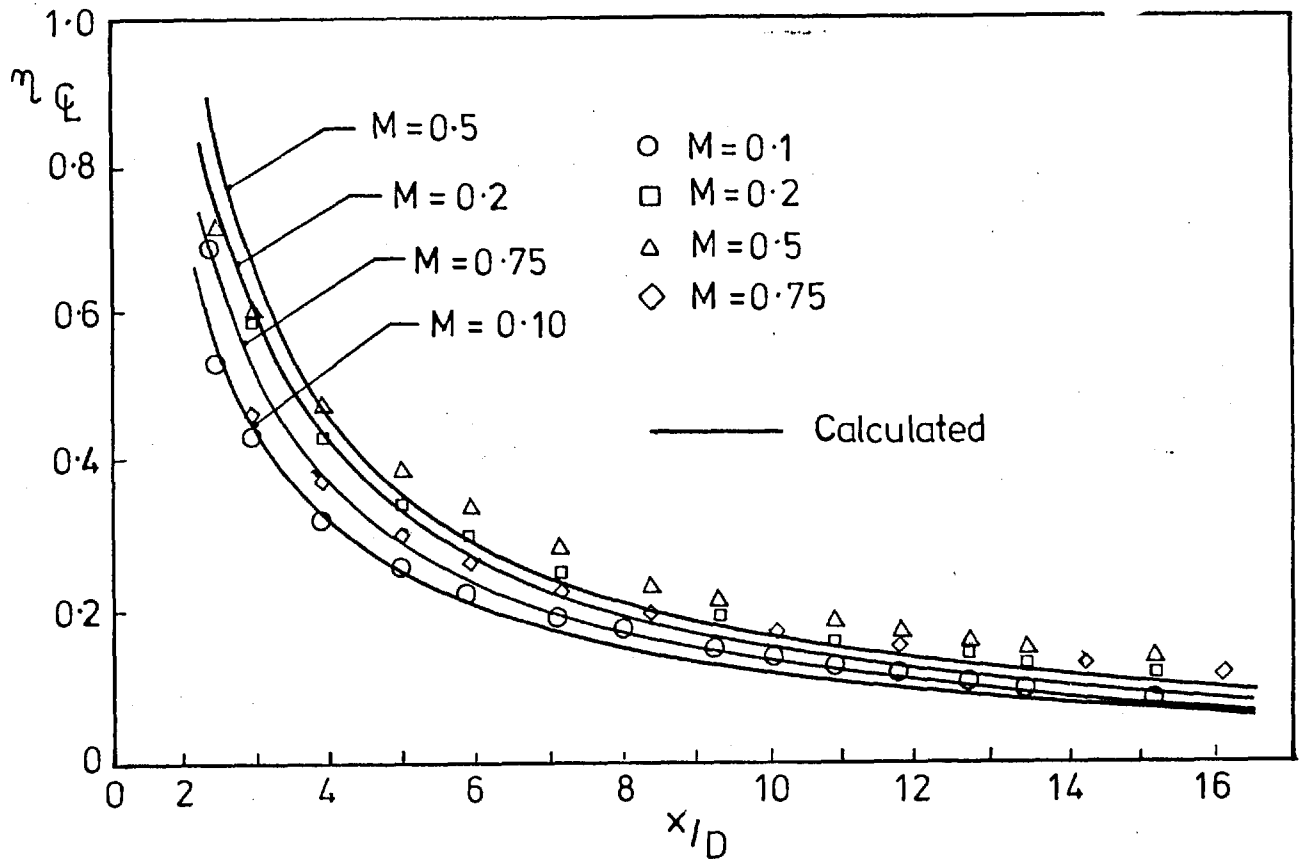


Fig. 7 : Predictions by the point source model (taken from (14))

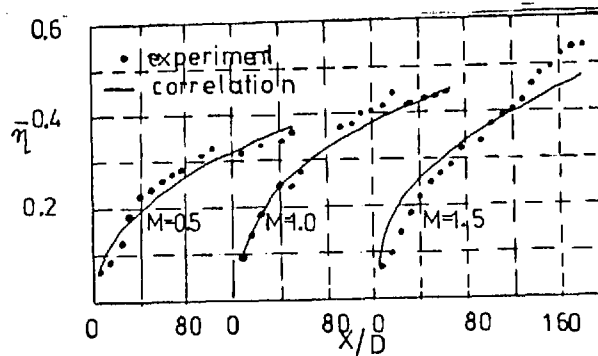


Fig. 8 : Predictions by the point source model

(taken from (16))

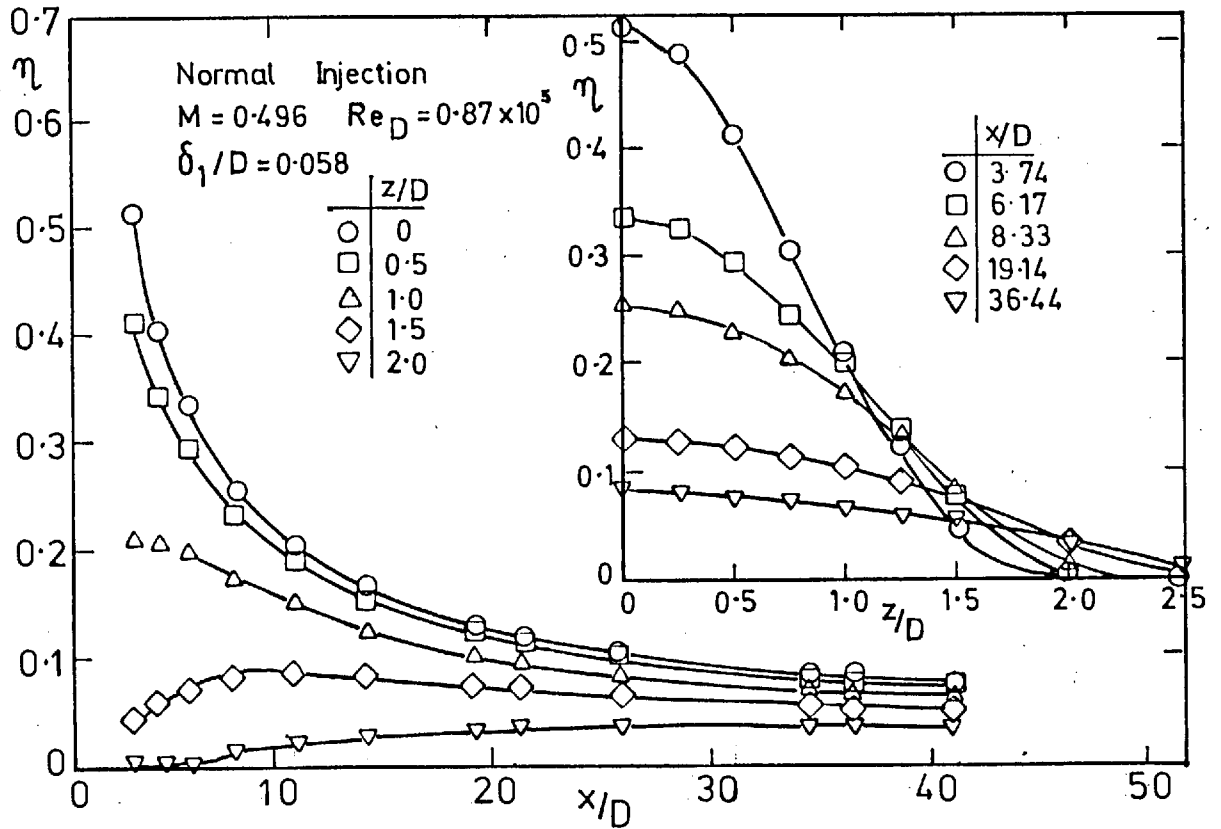


Fig. 9a : Effectiveness for $M=0.496$, single hole,
 $\alpha=90^\circ$ (taken from (29))

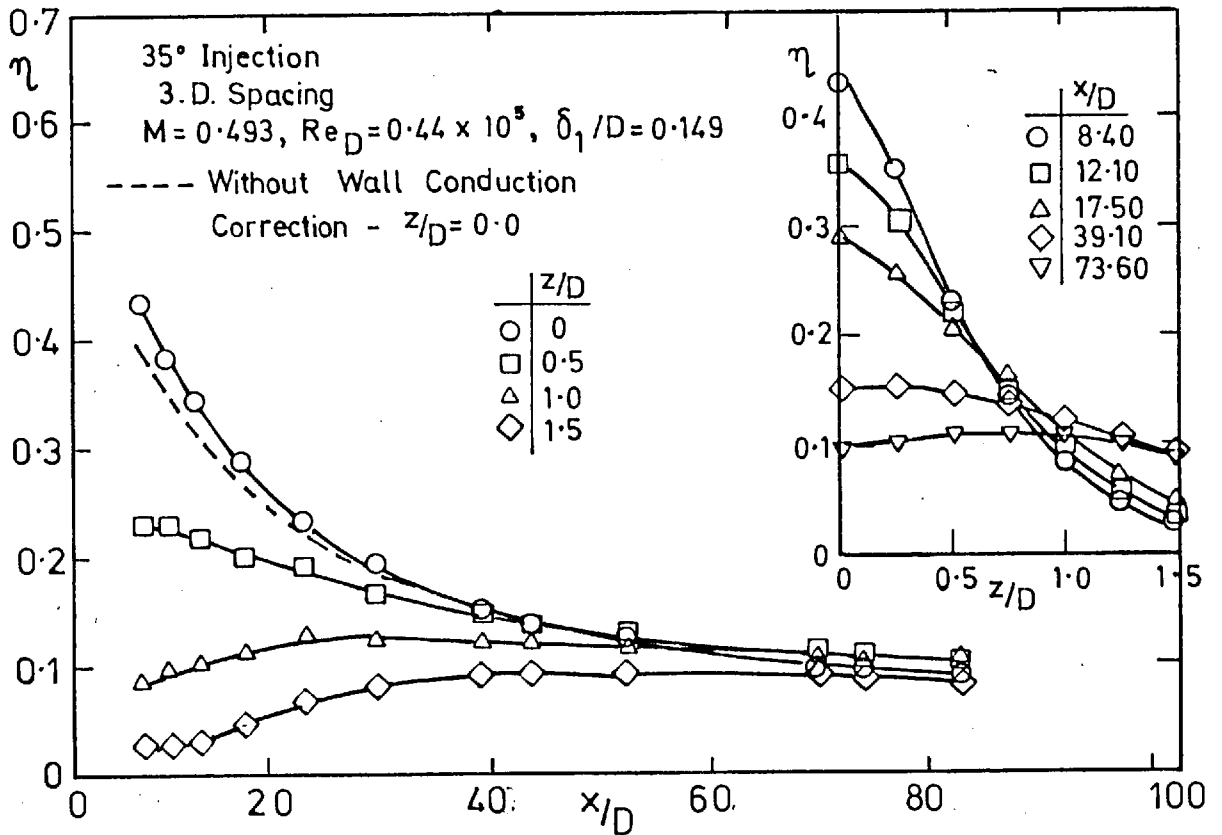


Fig. 9b : Effectiveness for $M=0.493$, single hole,
 $\alpha=35^\circ$ (taken from (29))

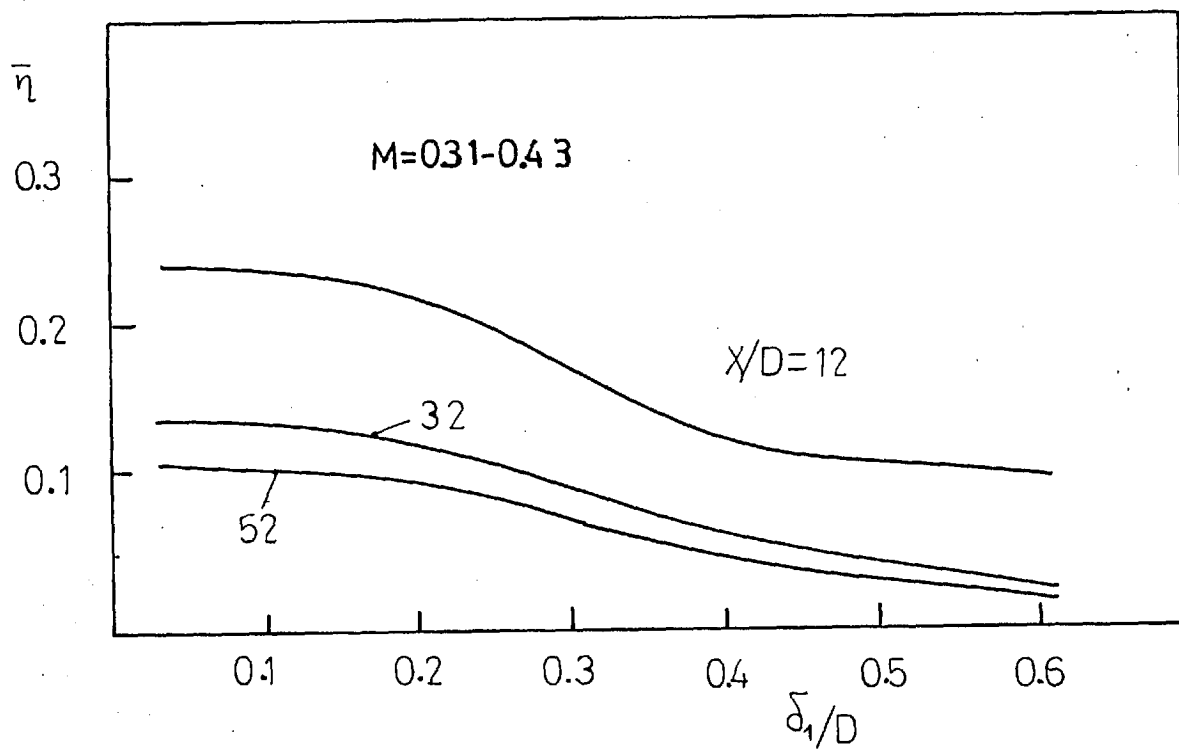


Fig. 10 : Effect of $\delta l/D$ on $\bar{\eta}$ (taken from (20))

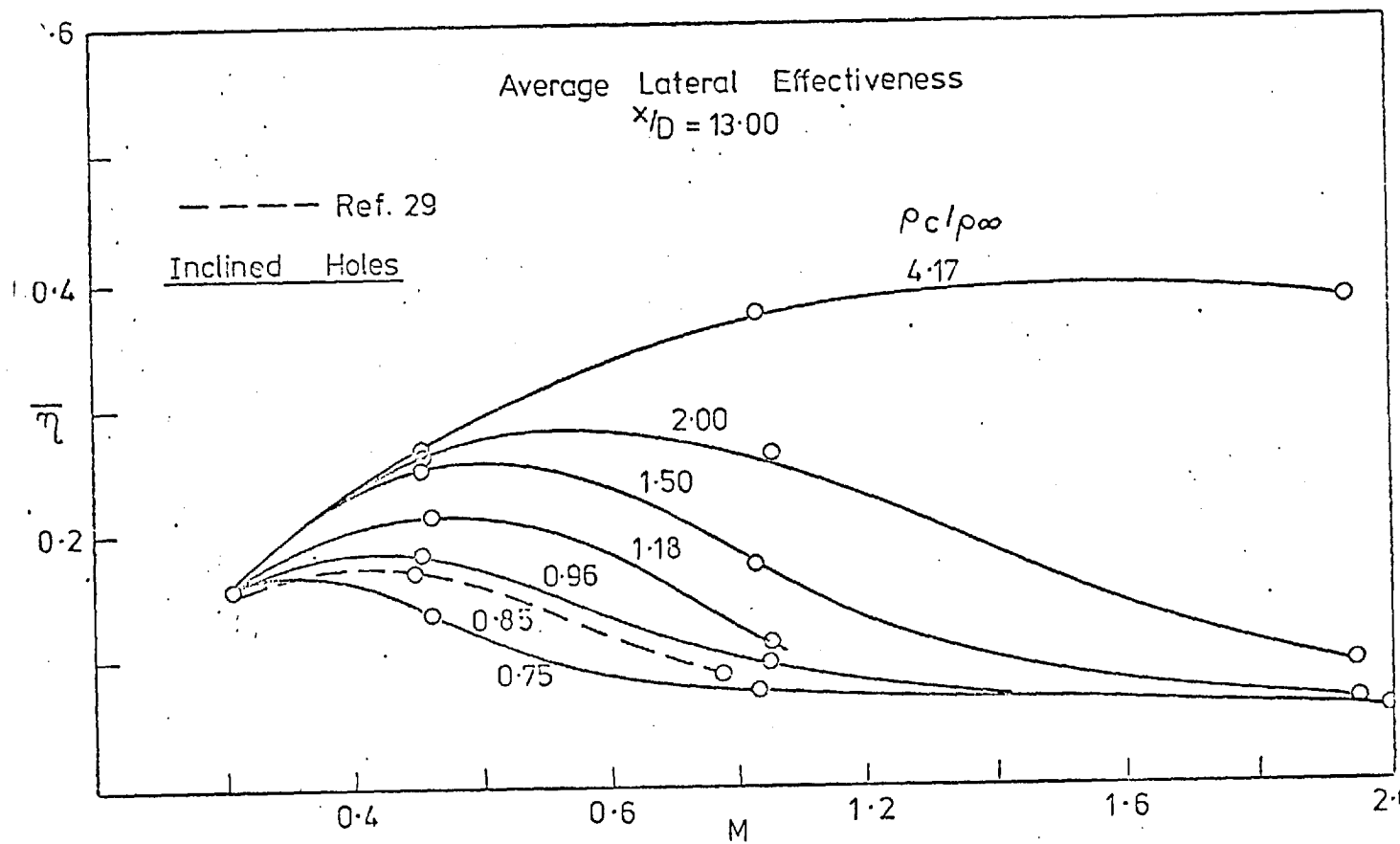


Fig. 11 : Effect of density ratio on $\bar{\eta}$ (taken from (18))

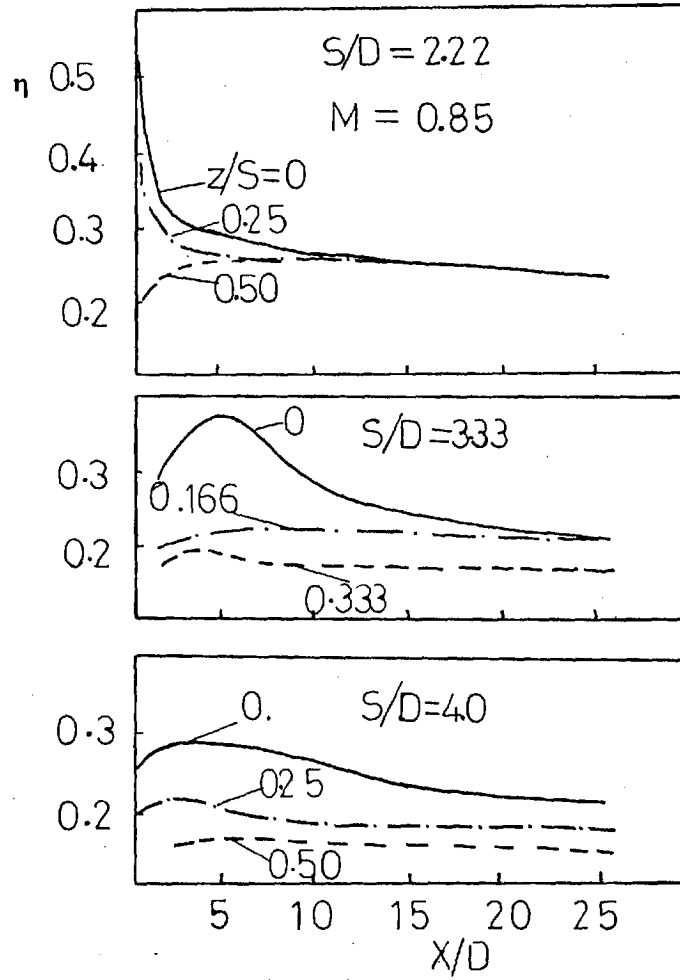


Fig. 12 : Effect of s/D on η (taken from (21)).

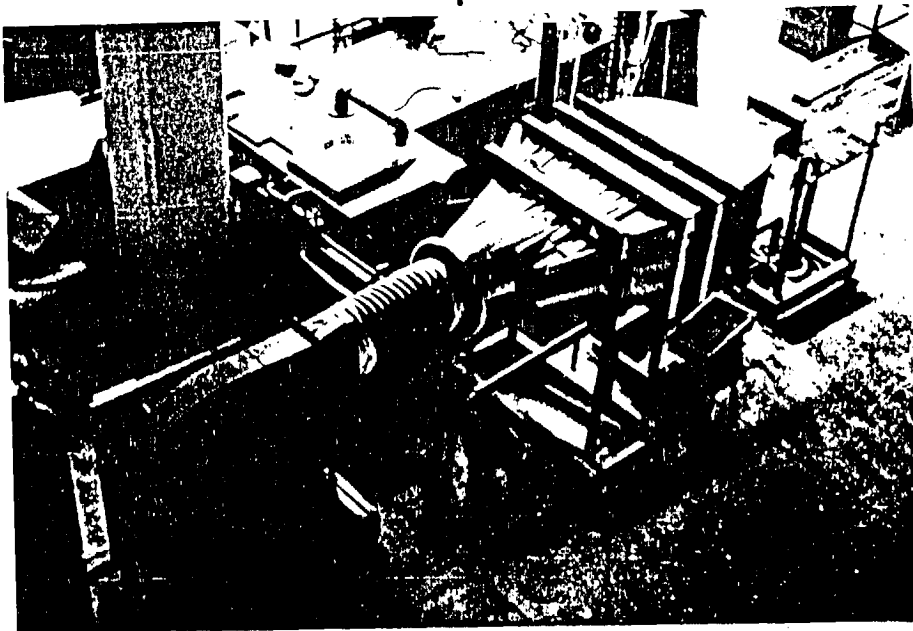


Fig. 13 : The open circuit wind tunnel.

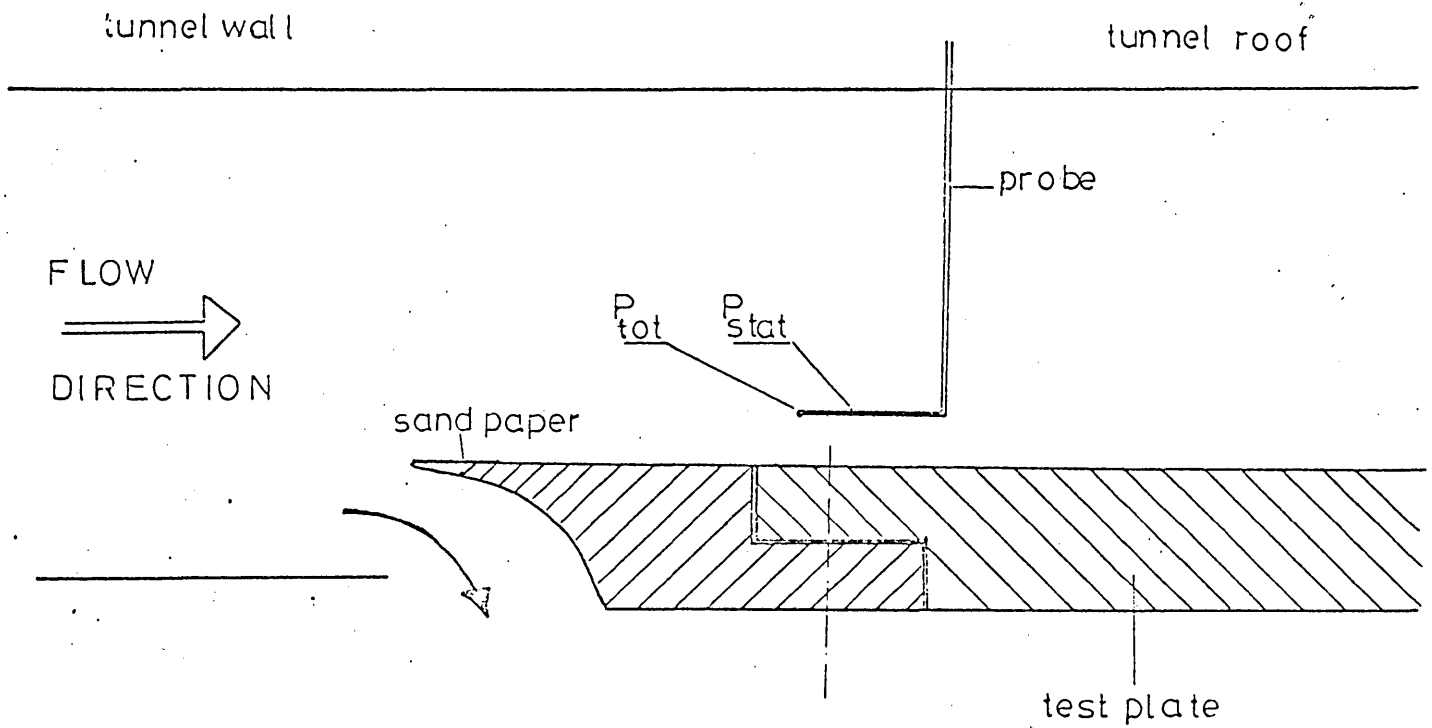


Fig. 14 : The entrance to the test section.

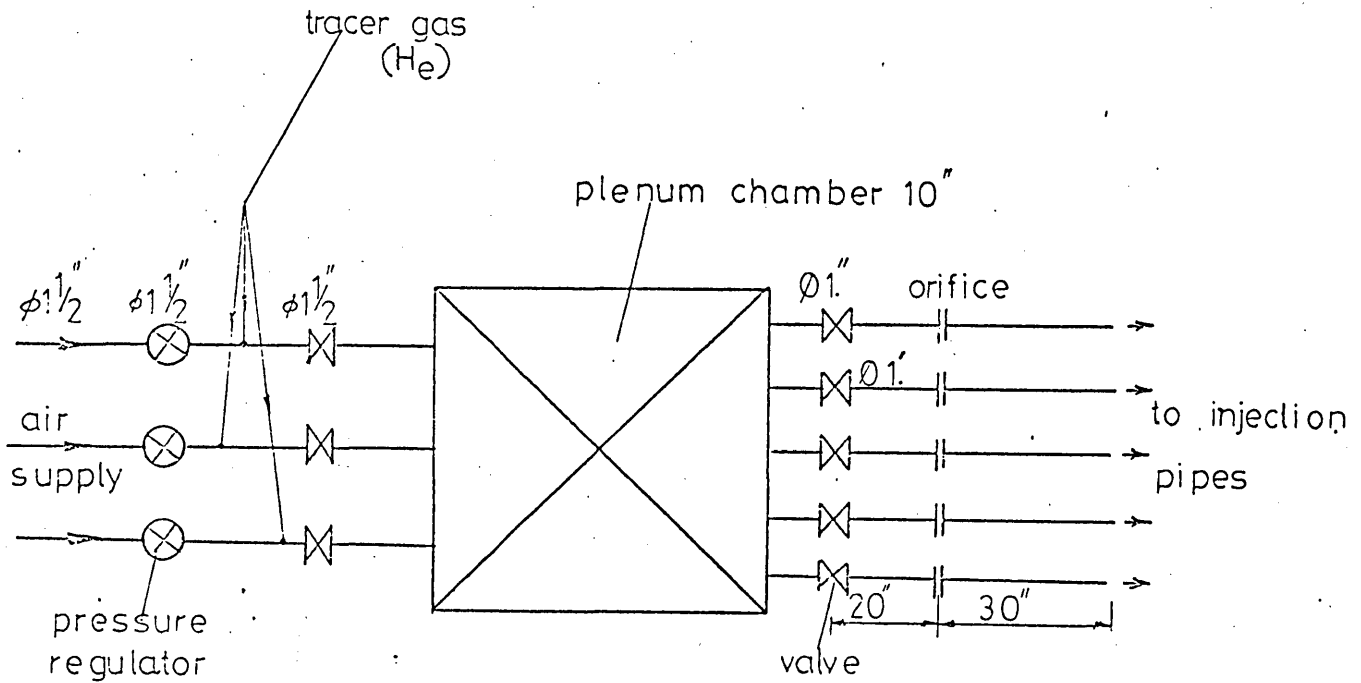


Fig. 15 : The secondary air distribution system.

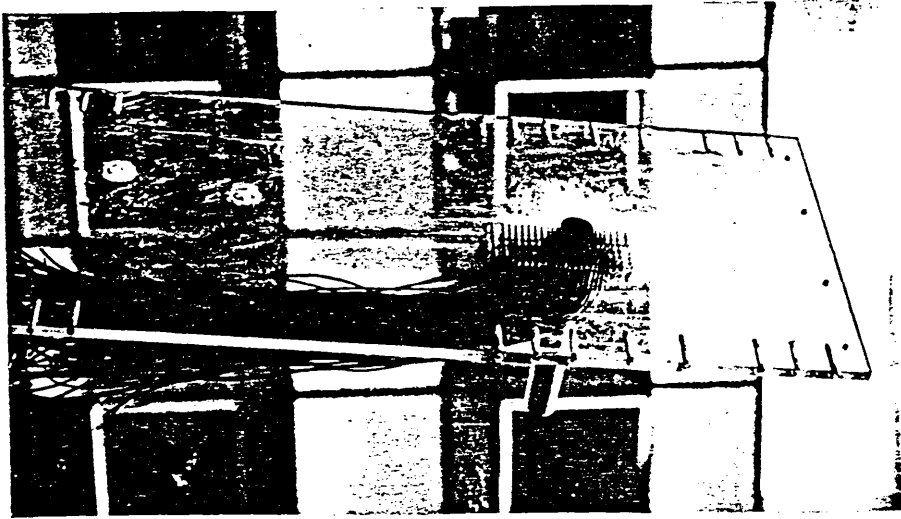


Fig. 16a.: The test plate for single hole, $\alpha=90^\circ$, study.



Fig. 16c : The test plate for the double row, $\alpha=30^\circ$, study.

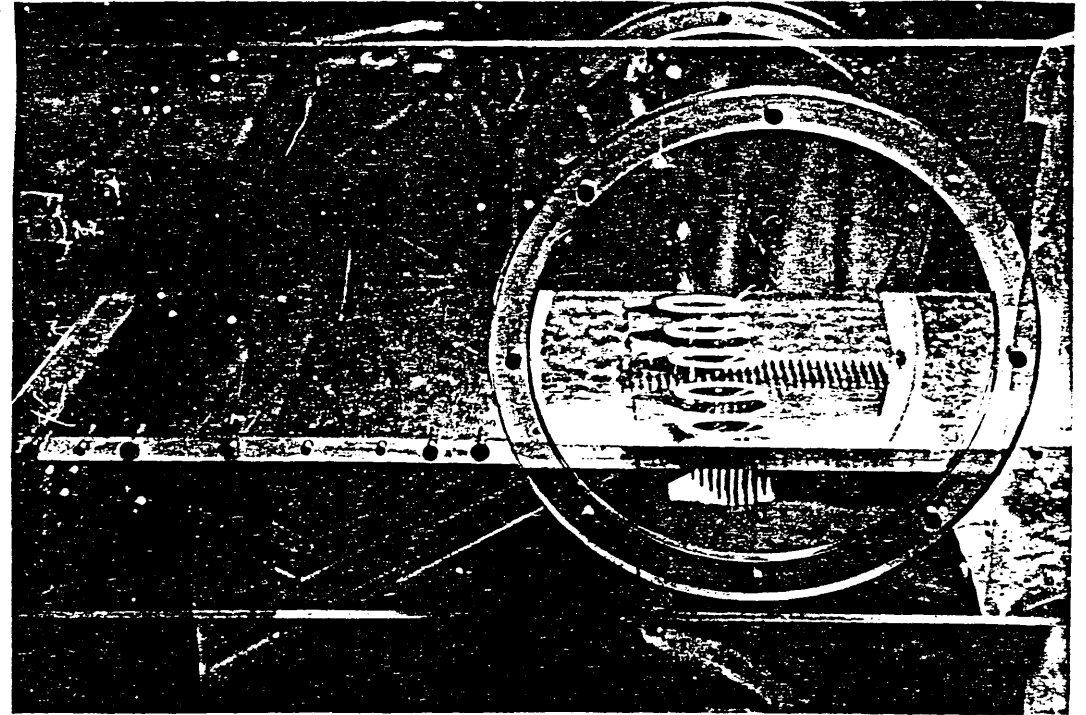


Fig. 16b : The test plate for the single hole,
single row, $\alpha=30^\circ$, study.

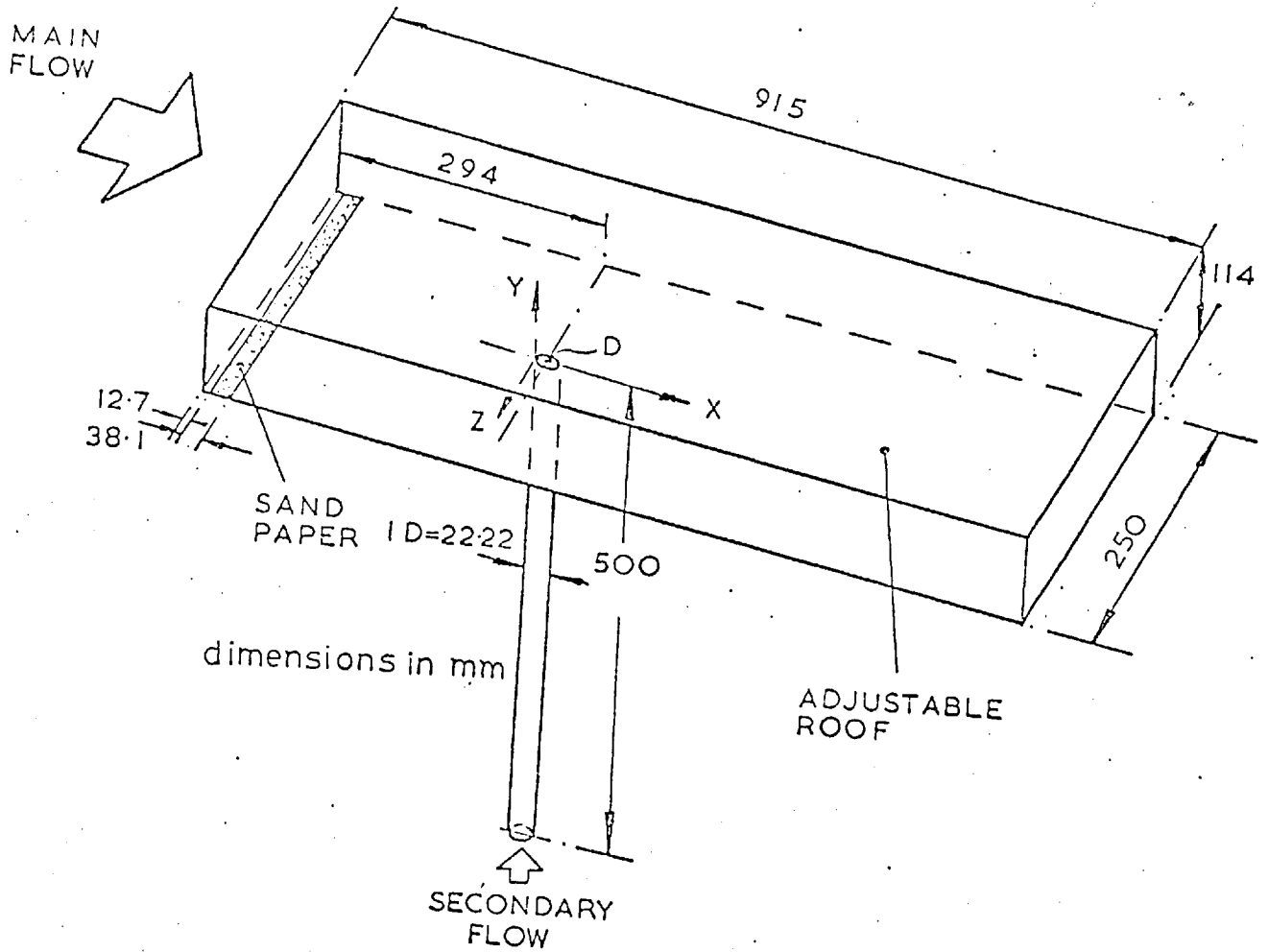
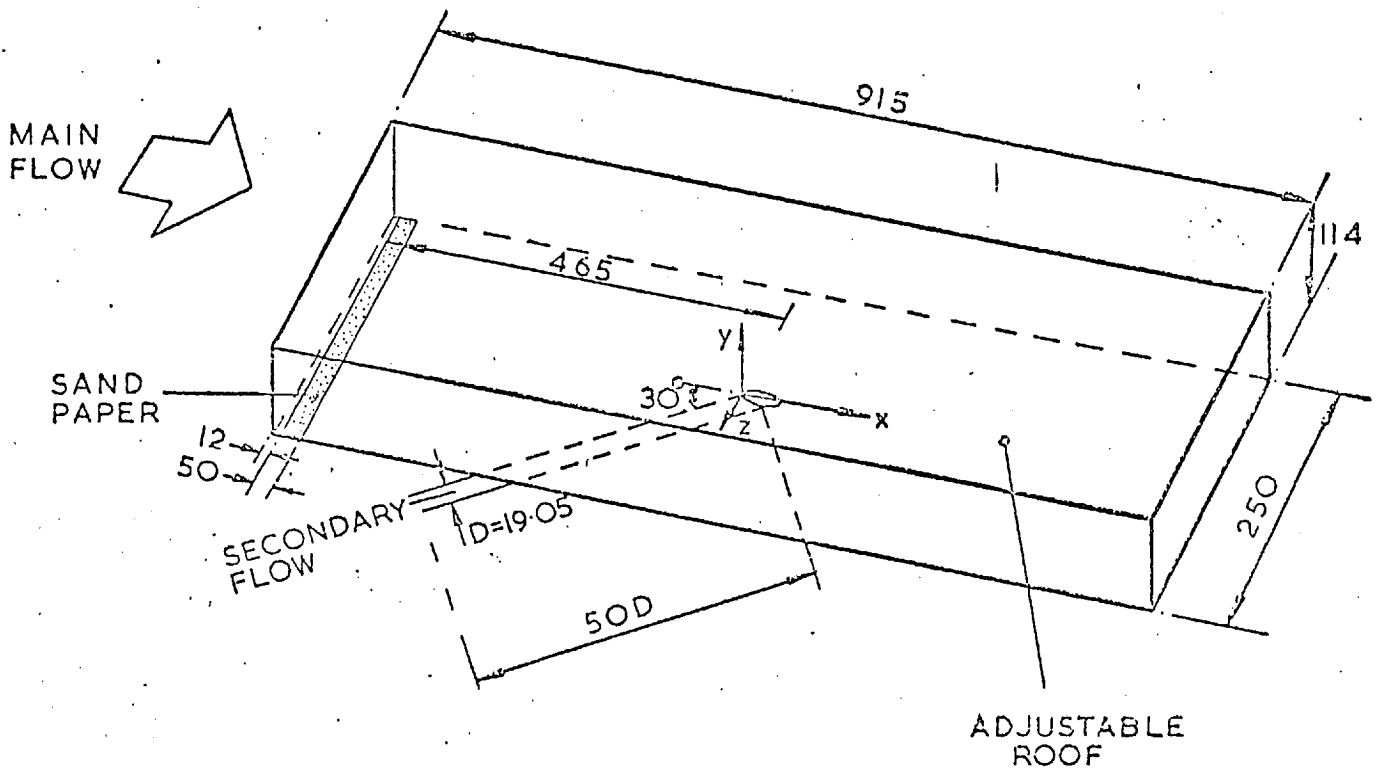


Fig. 17a : Dimensions of the test section for $\alpha=90^\circ$, single hole, study.



DIMENSIONS IN mm

Fig. 17b : Dimensions of the test section for $\alpha=30^\circ$, single hole, study.

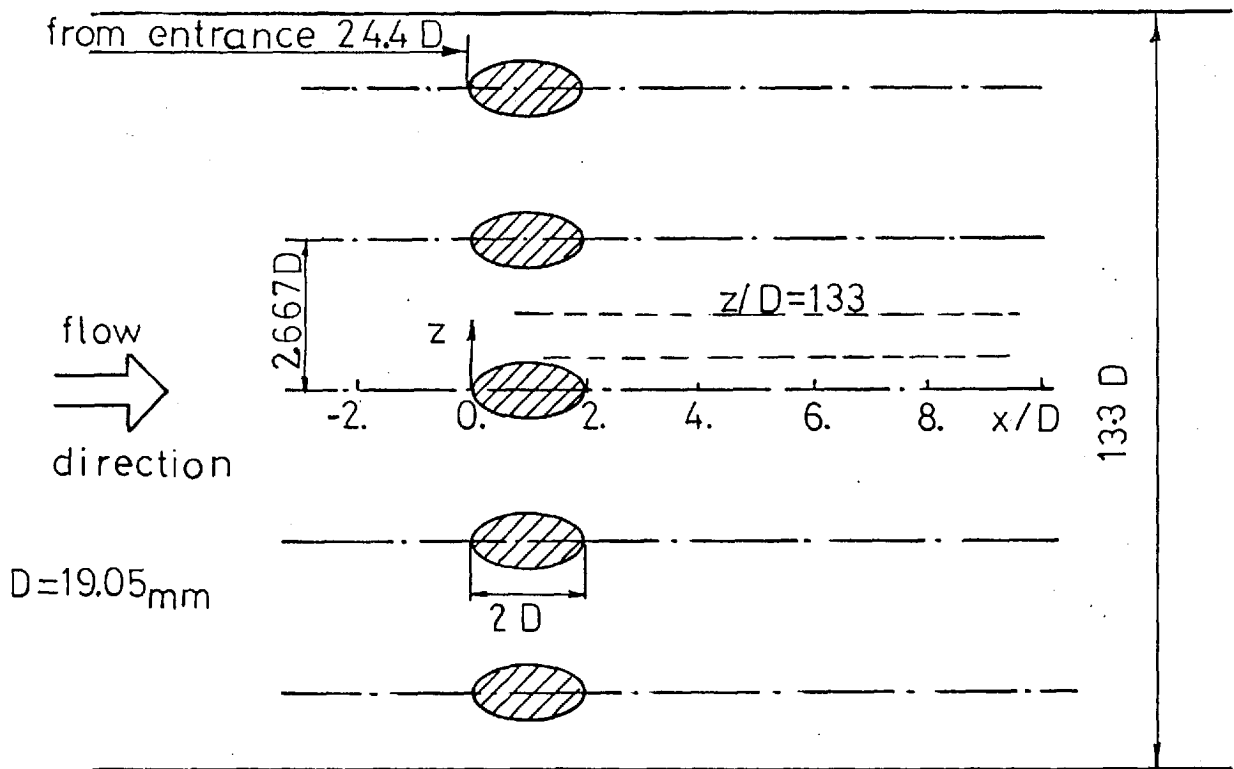


Fig. 17c : Dimensions of the test section for
 $\alpha=30^\circ$, single row, study.

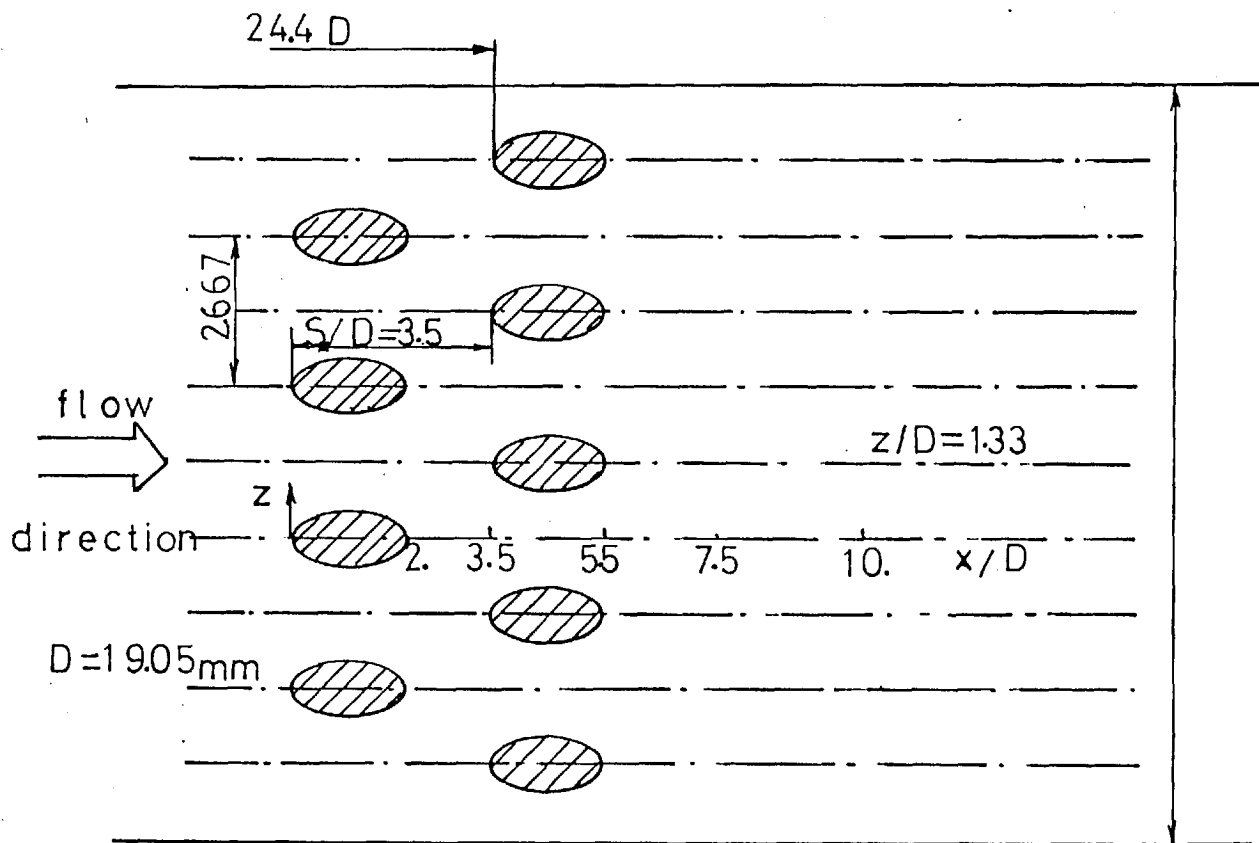


Fig. 17d : Dimensions of the test section for
 $\alpha=30^\circ$, double row, study.

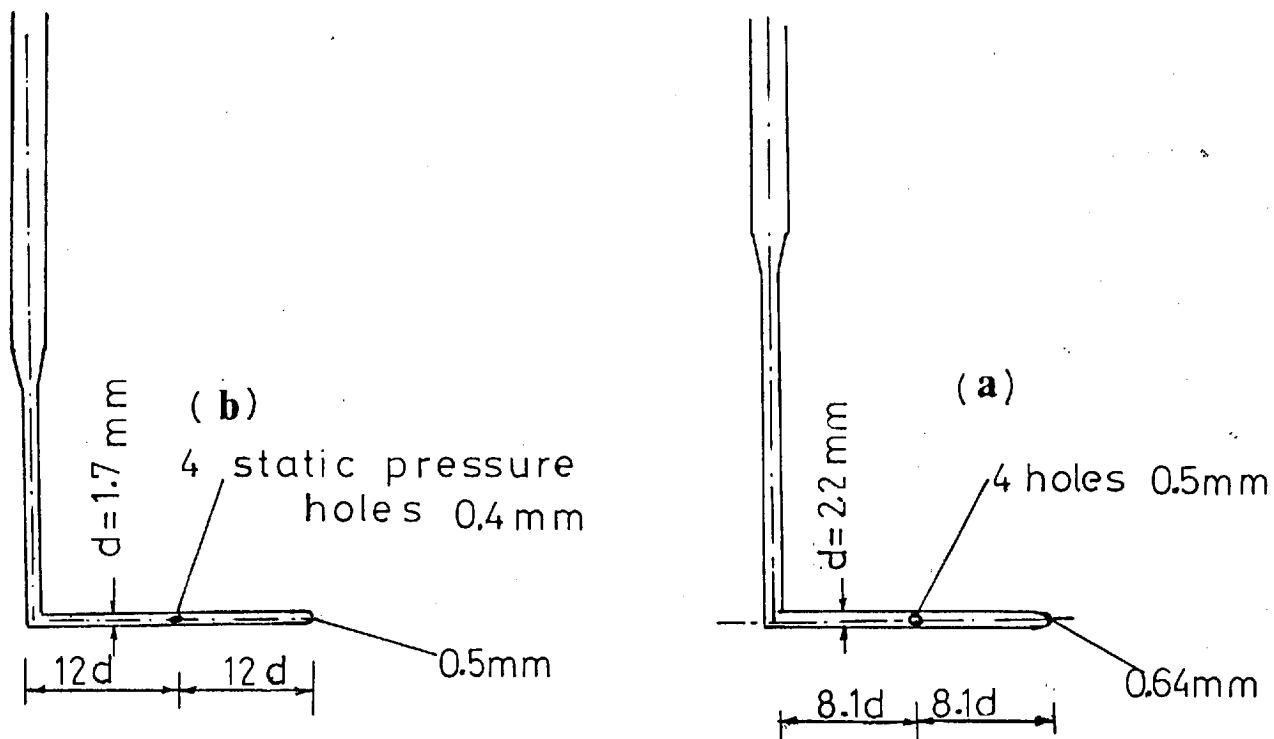


Fig. 18 : Dimensions of the Pilot-static probes.

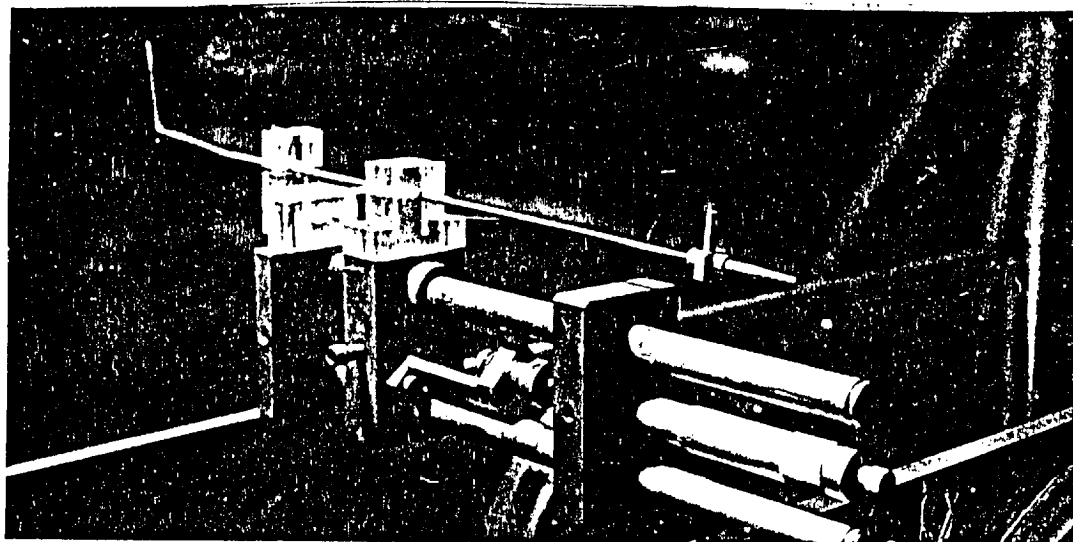


Fig. 18c : The traversing mechanism.

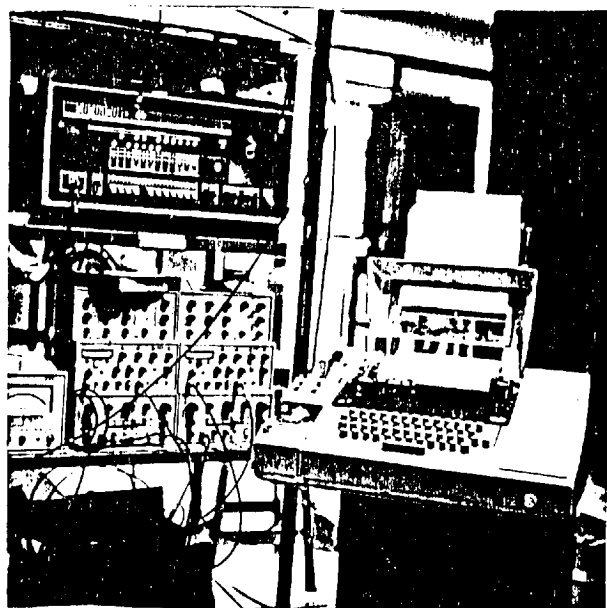


Fig. 18d : The hot-wire anemometry set up.

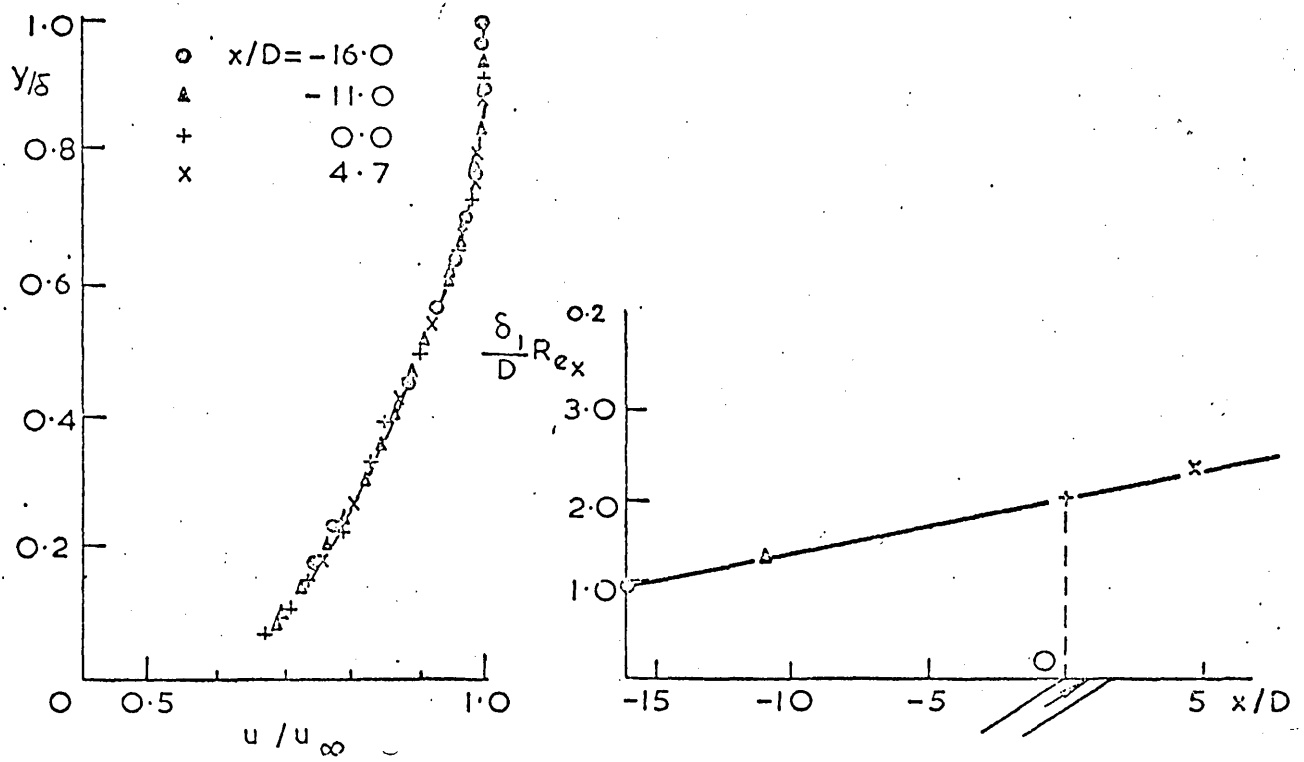


Fig. 19a : Development of the velocity profile Fig. 19b : Development of the boundary layer

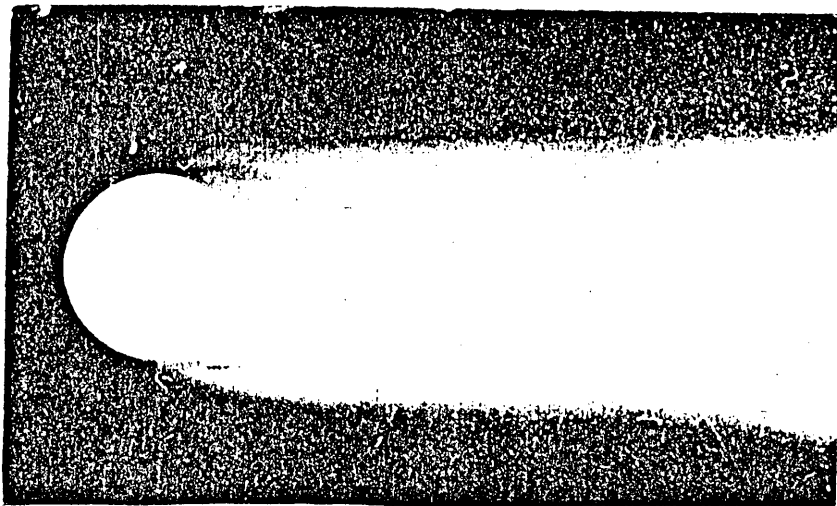


Fig. 20a : Lateral spread of the jet ($M=0.24$).



Fig. 20b : Vertical spread of the jet ($M=0.24$).

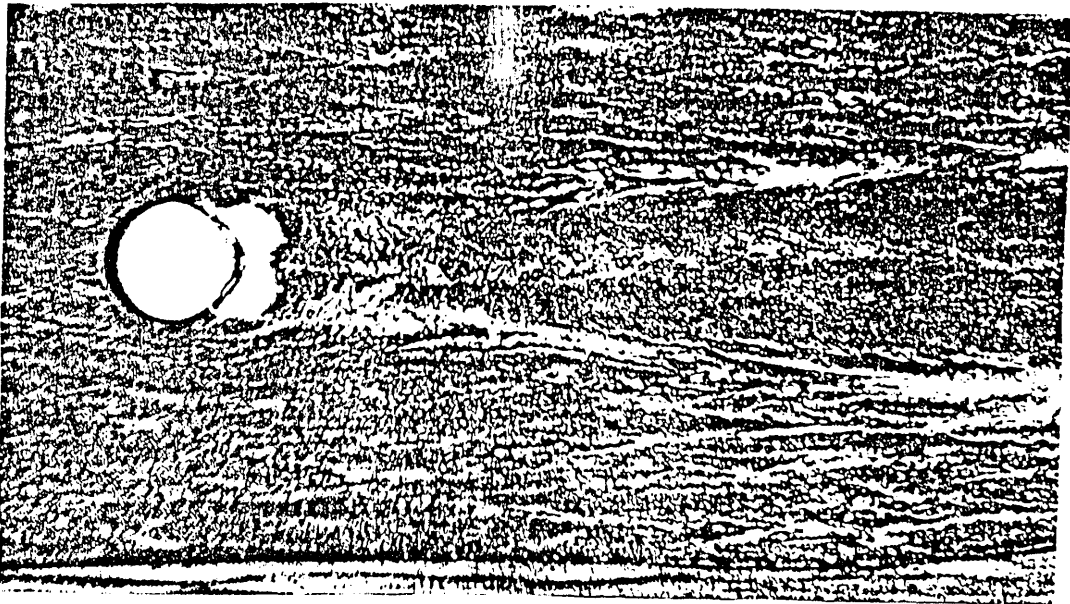


Fig. 21a : Surface flow pattern ($M=0.10$)



Fig. 21b : Surface flow pattern ($M=0.24$).

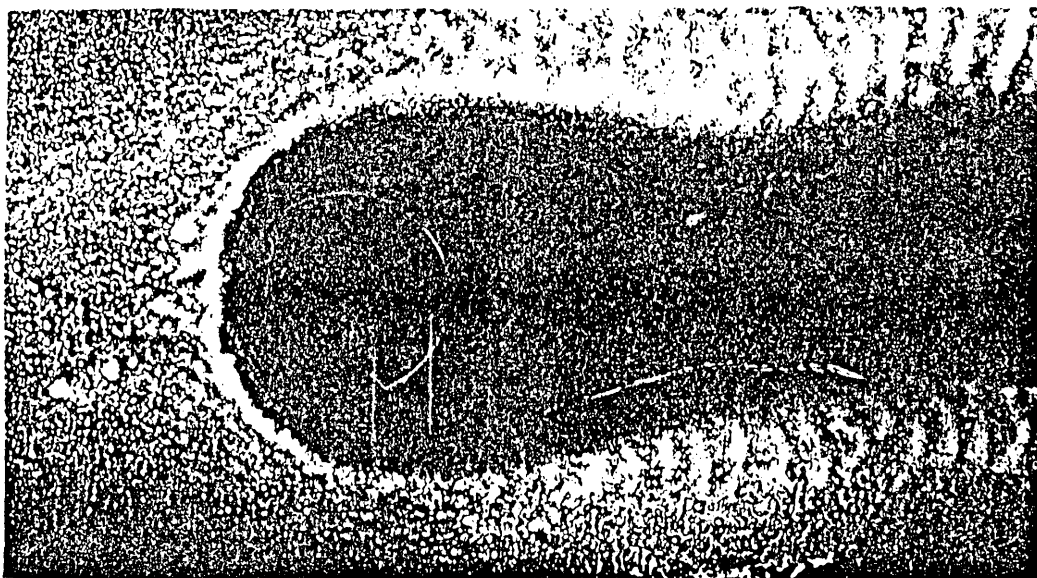


Fig. 21c : Surface flow pattern ($M=0.24$).

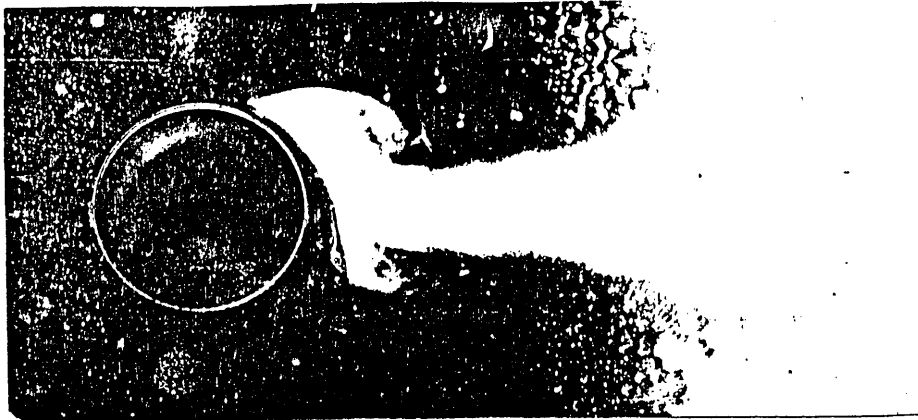


Fig. 2ld : Region of reverse flow (intermediate time).

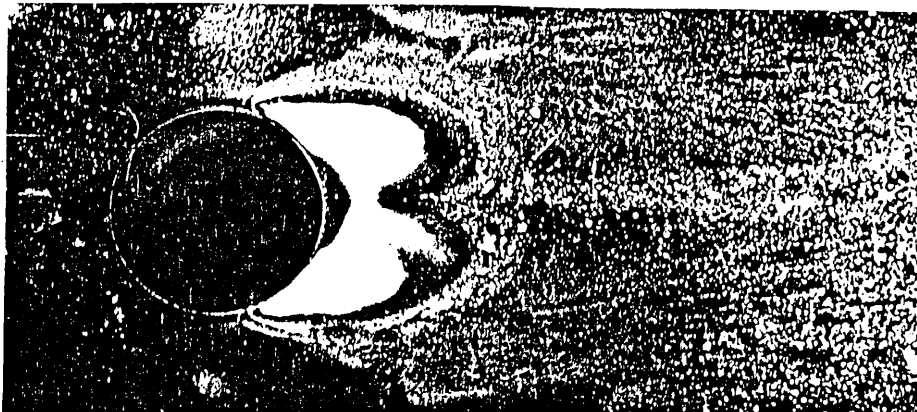
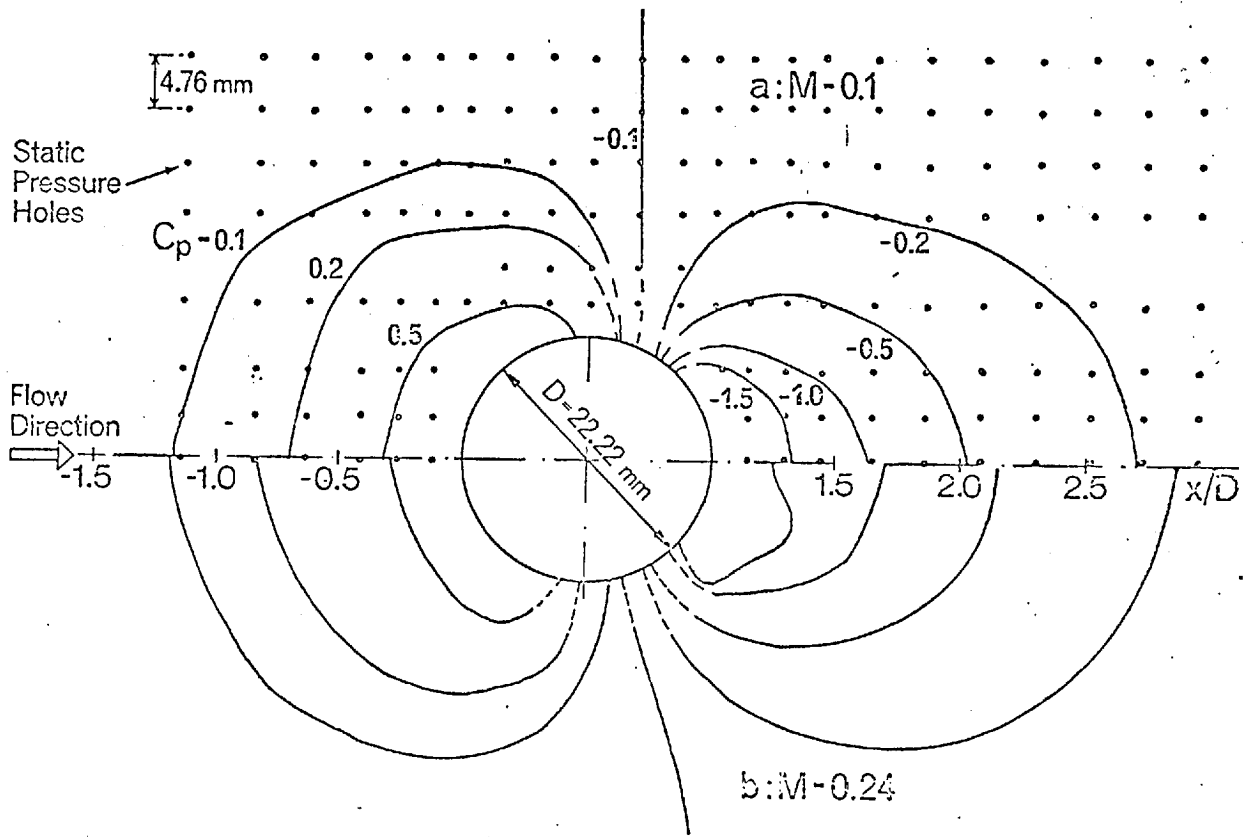


Fig. 2le : Region of reverse flow (final pattern).



(c) $M = 1.0$

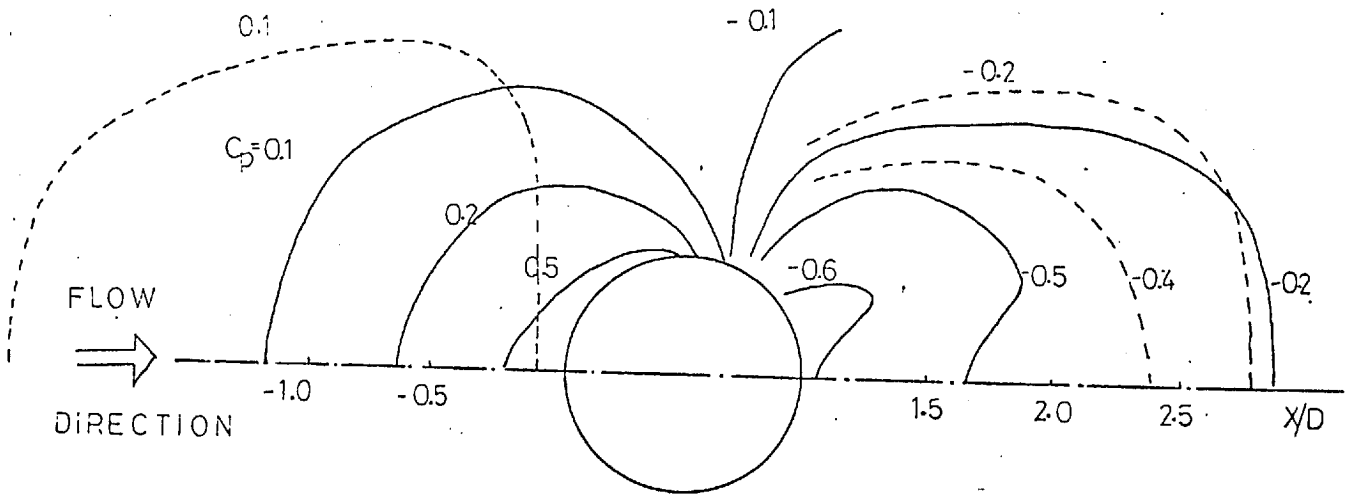


Fig. 22 : Surface isobaric contours.

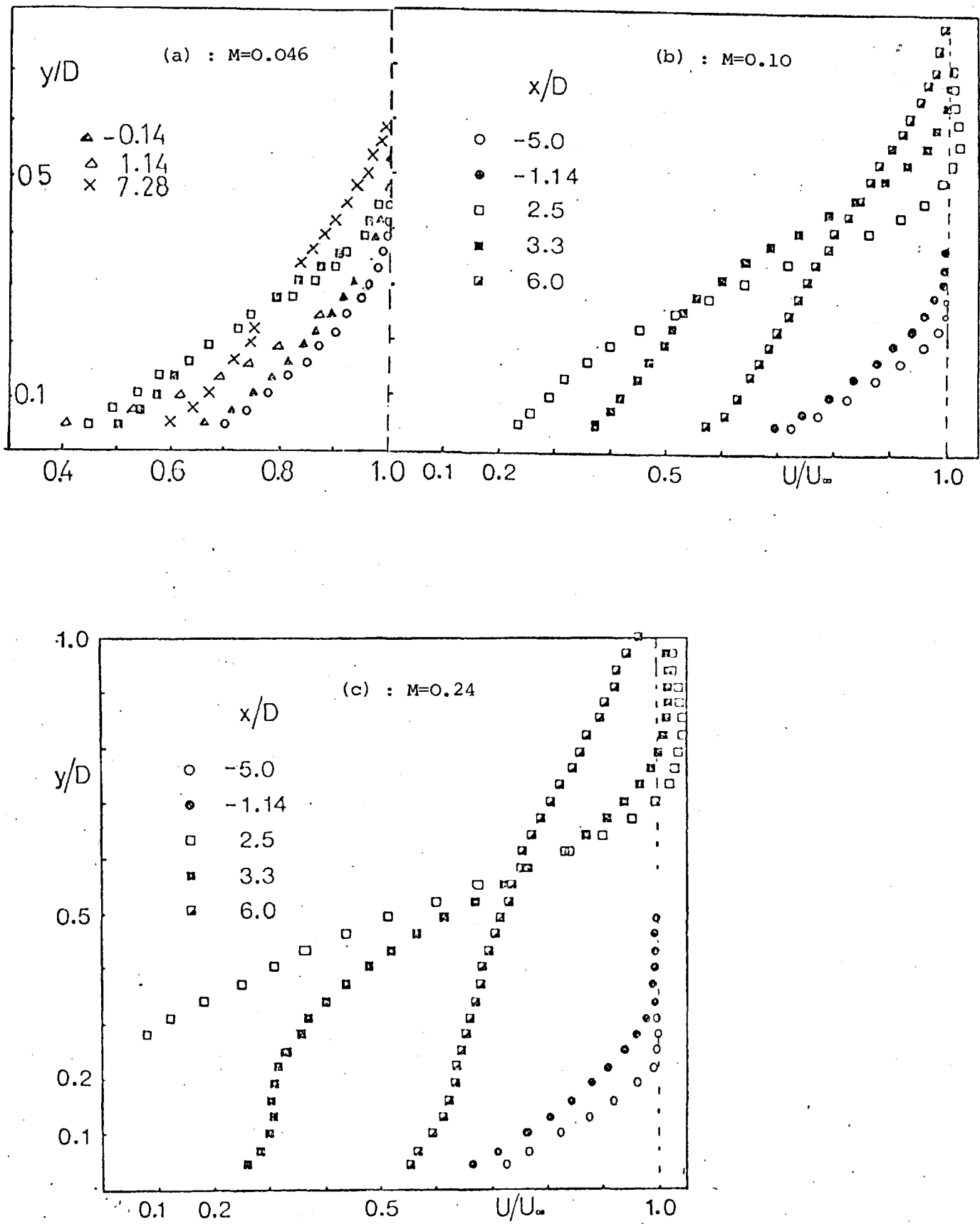


Fig. 23 : Profiles of streamwise velocity along the
centre line

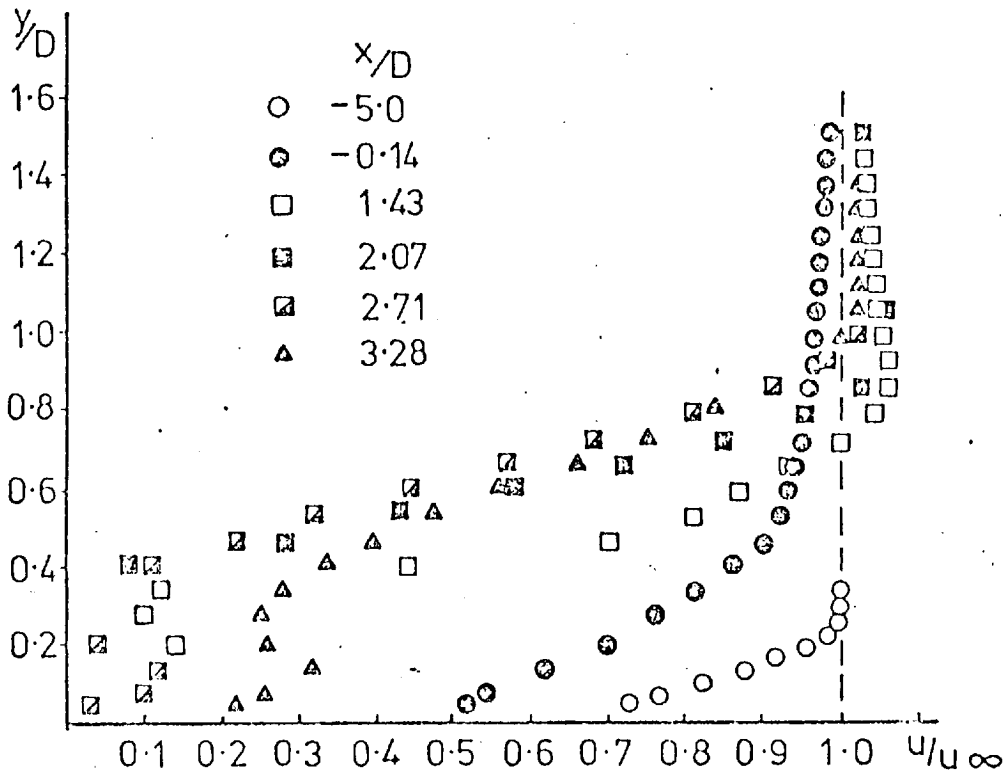


Fig. 23d : Profiles of streamwise velocity, $z/D=0.0$,

$\alpha=90^\circ$, $M=0.50$.

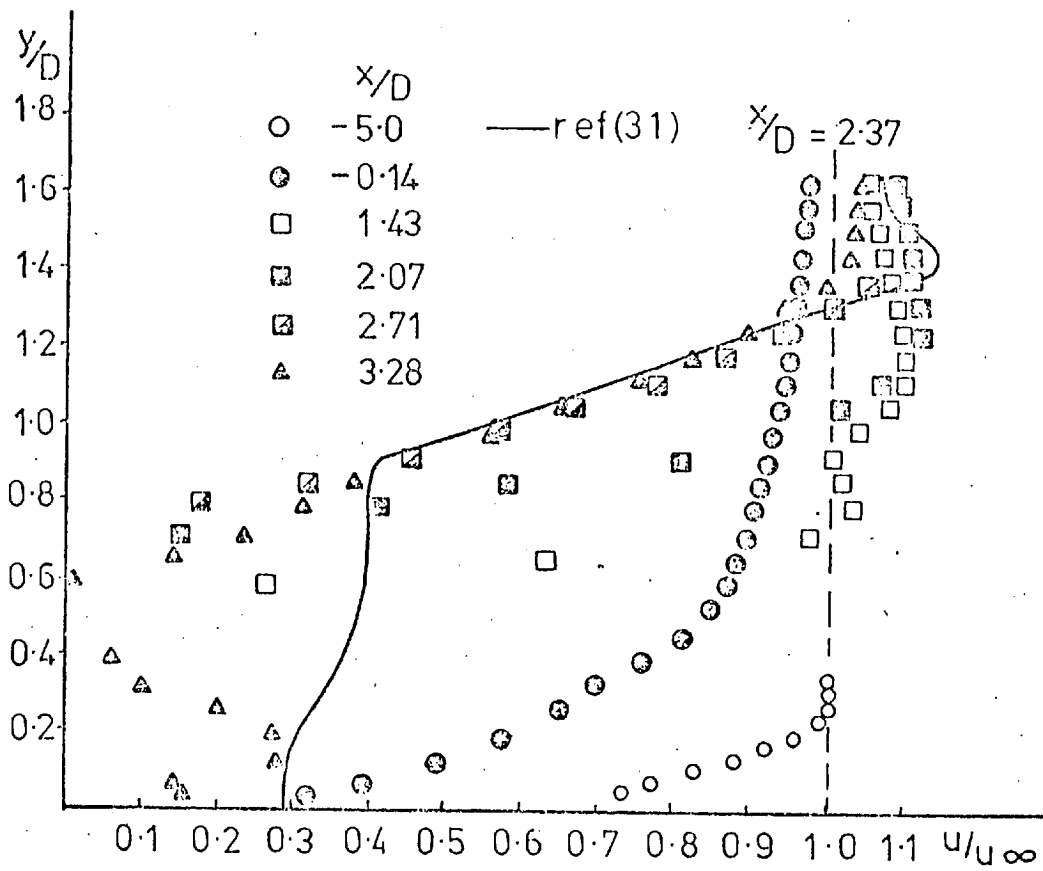


Fig. 23e : Profiles of streamwise velocity, $z/D=0.0$,

$\alpha=90^\circ$, $M=1.0$.

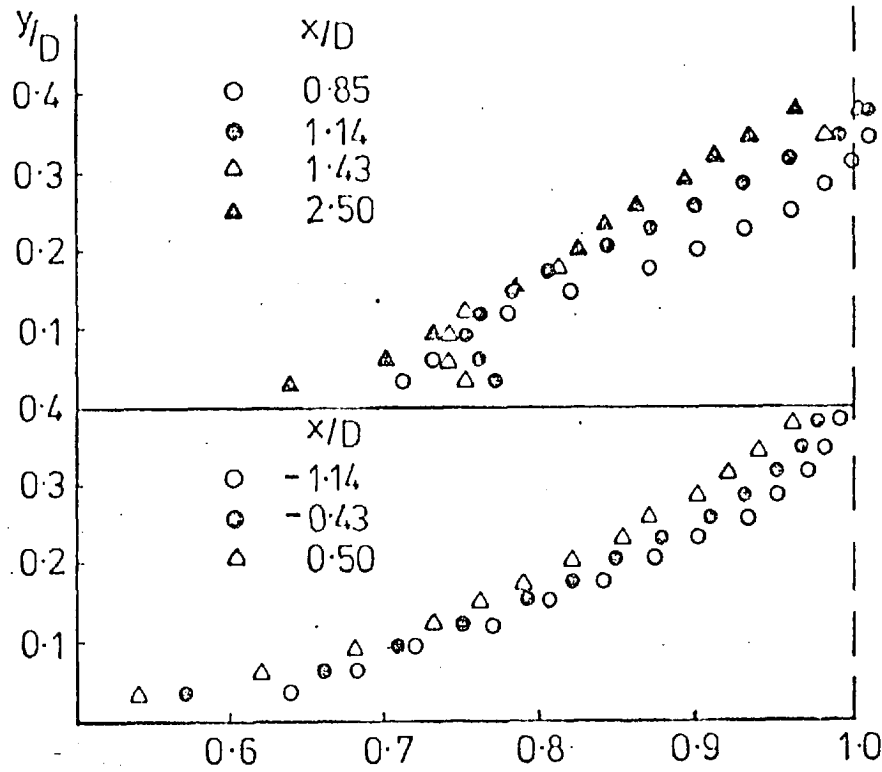


Fig. 24a : Profiles of streamwise velocity, $z/D=0.64$,
 $\alpha=90^\circ$, $M=0.24$.

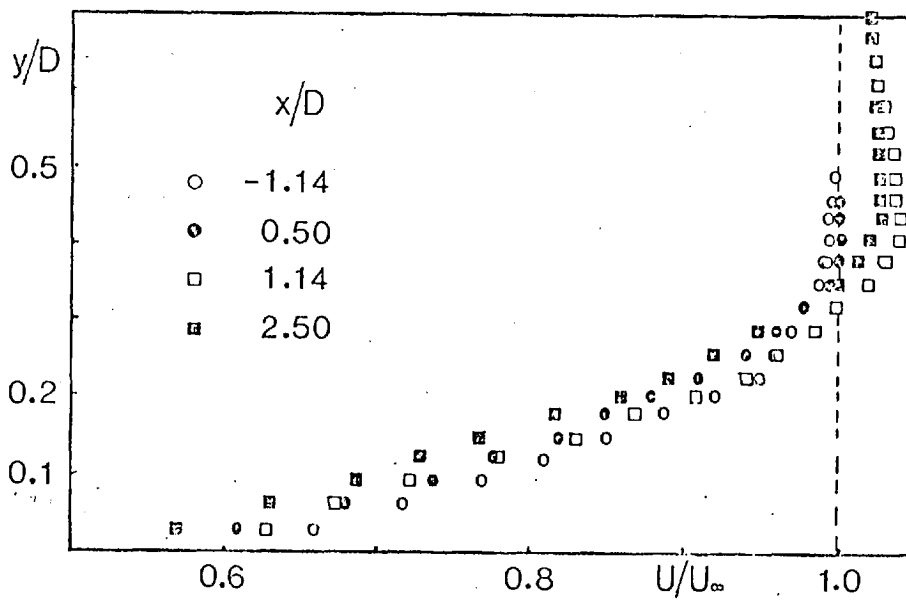


Fig. 24b : Profiles of streamwise velocity, $z/D=0.64$,
 $\alpha=90^\circ$, $M=0.10$.

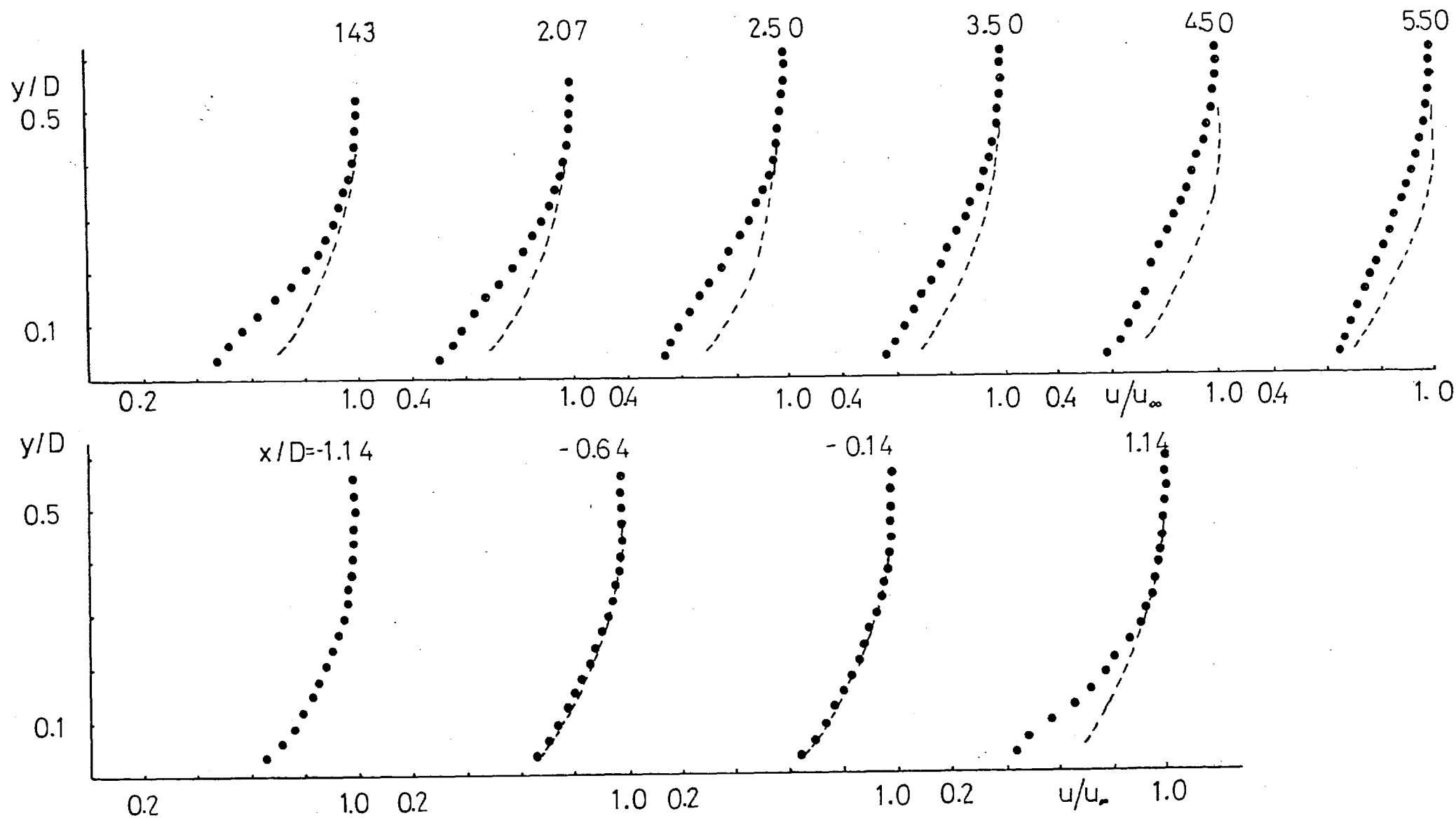


Fig. 24c : Profiles of streamwise velocity, $z/D=0.36$

$\alpha=90^\circ$, $M=0.046$.

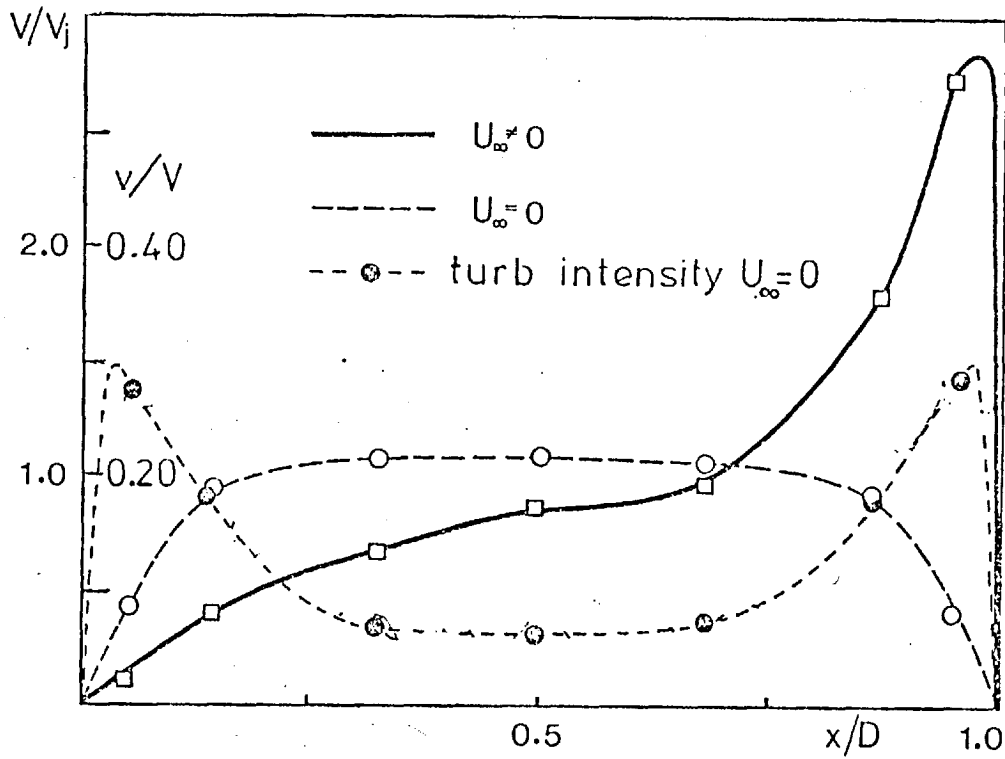


Fig. 25a : Exit velocity of secondary jet, $M=0.10$.

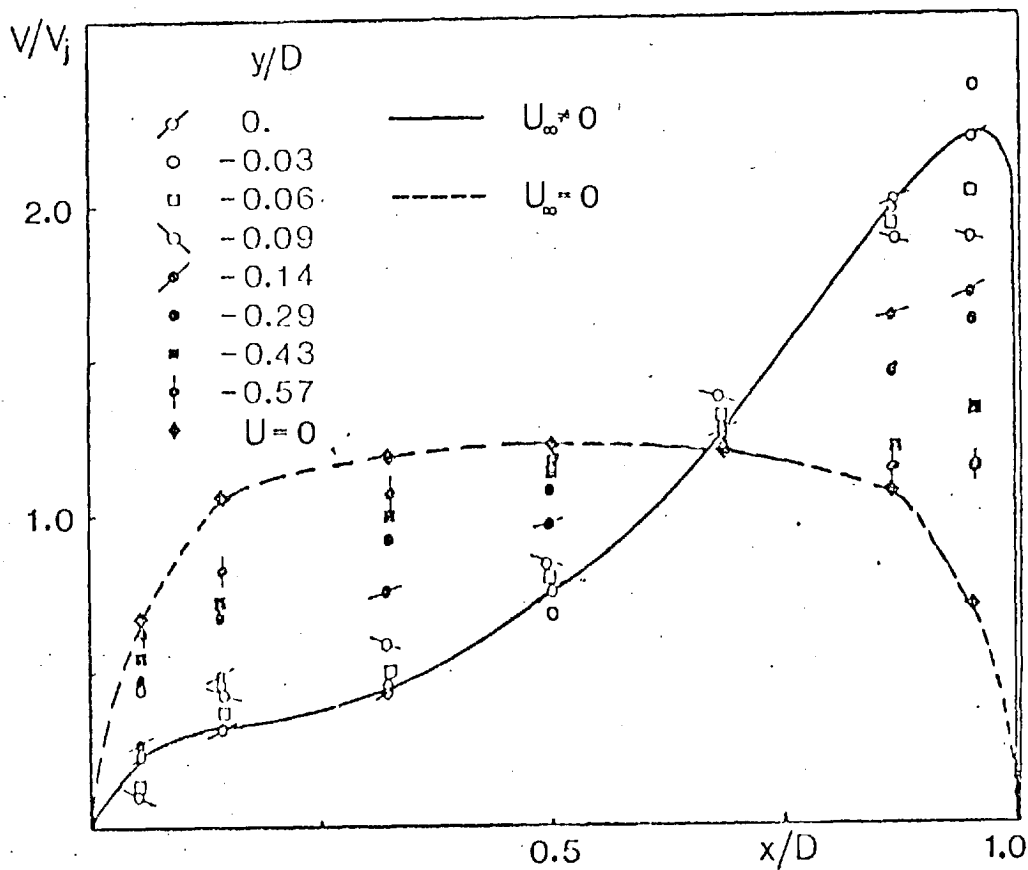


Fig. 25b : Velocity of secondary jet at and near exit plane, $M=0.24$.

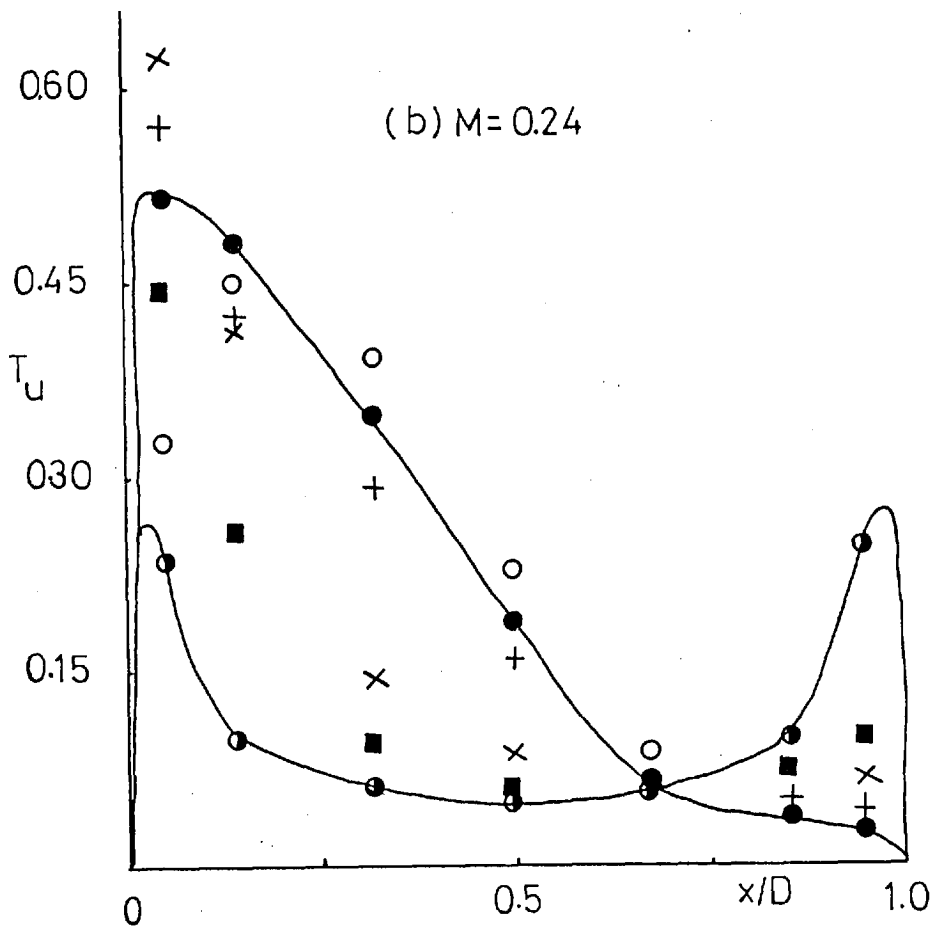
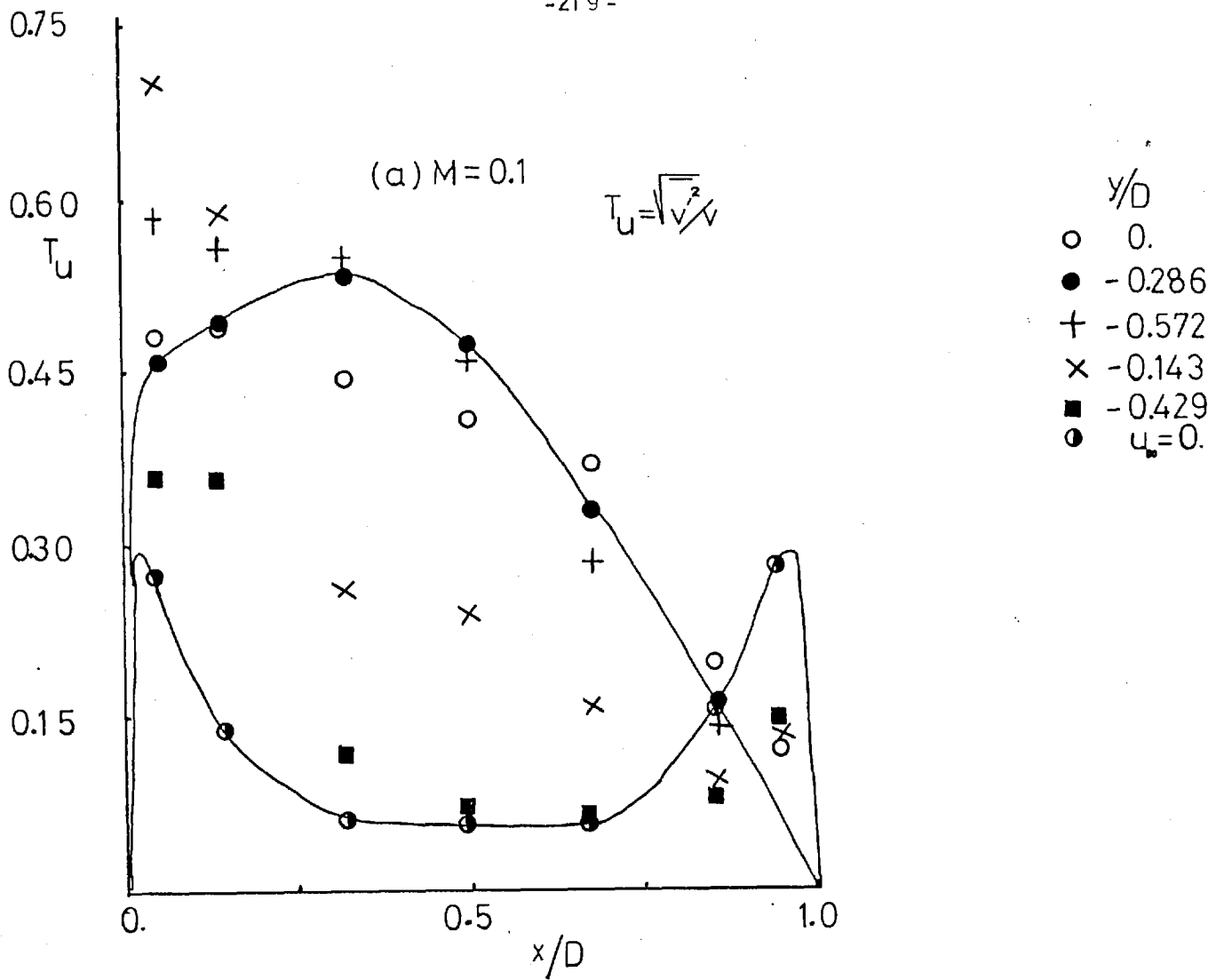


Fig. 26 : Turbulence intensity of secondary jet, $\alpha=90^\circ$

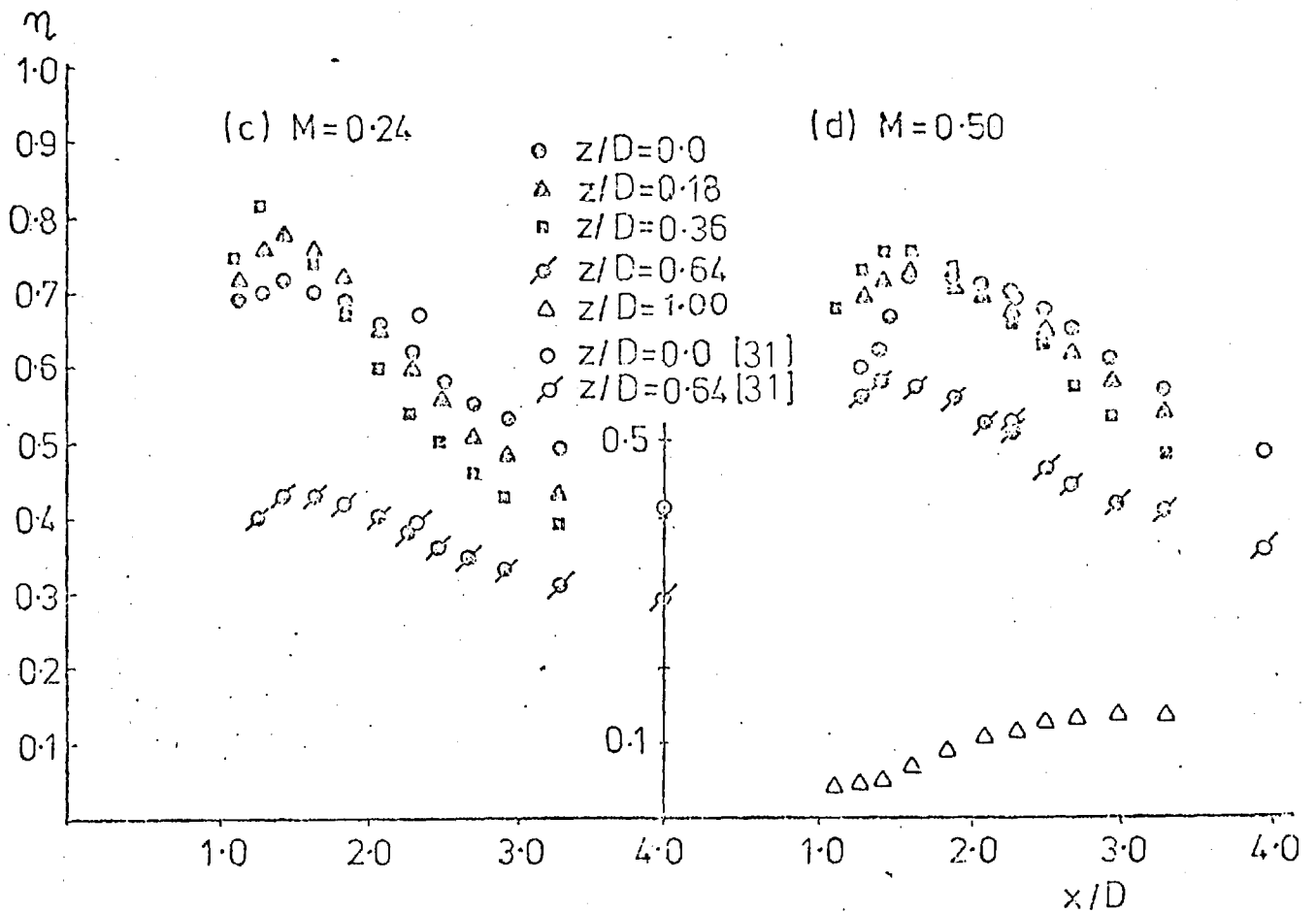
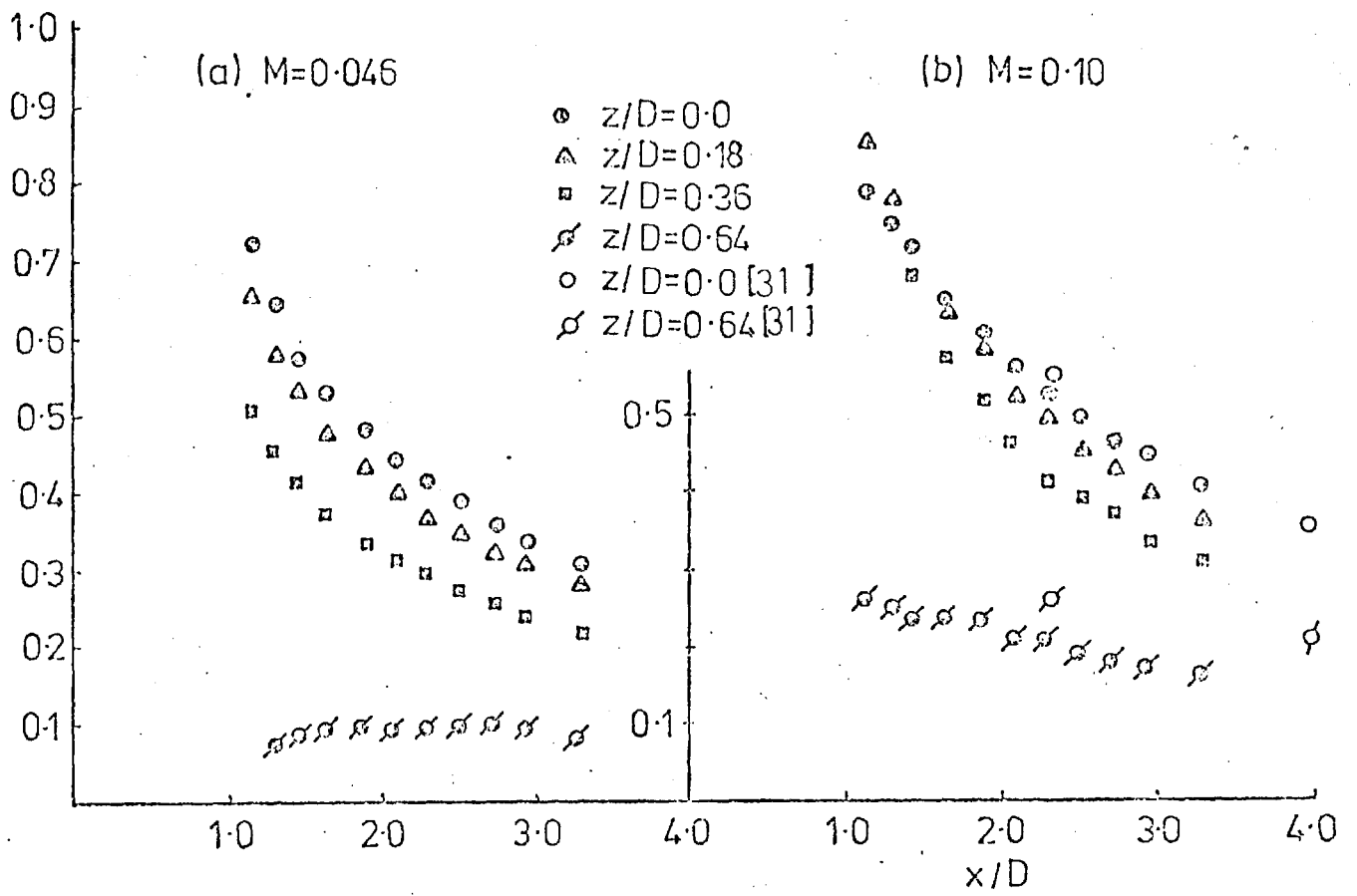


Fig. 27 : Wall film cooling effectiveness, $\alpha=90^\circ$.

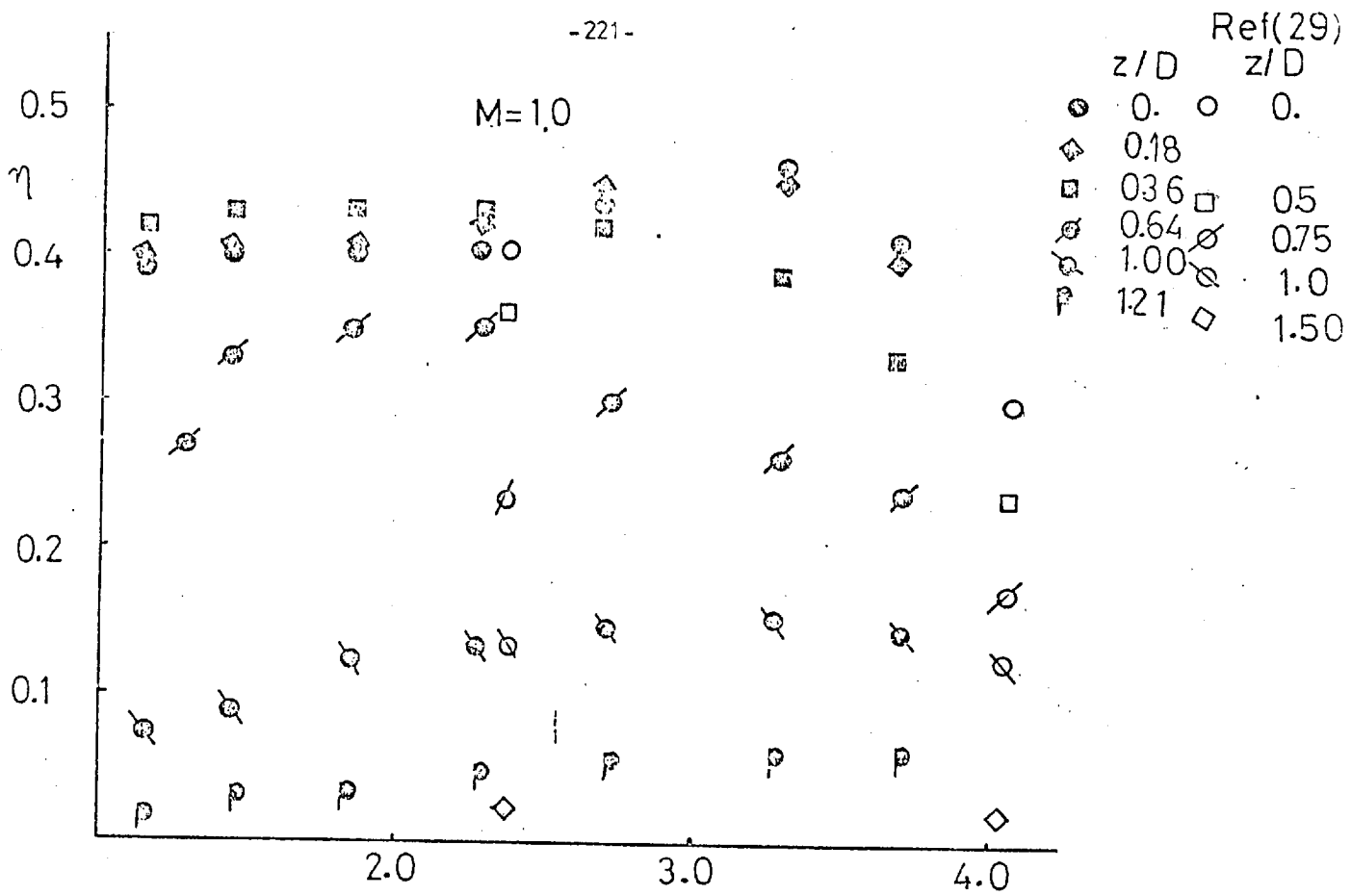


Fig. 27e : Wall film cooling effectiveness, x/D

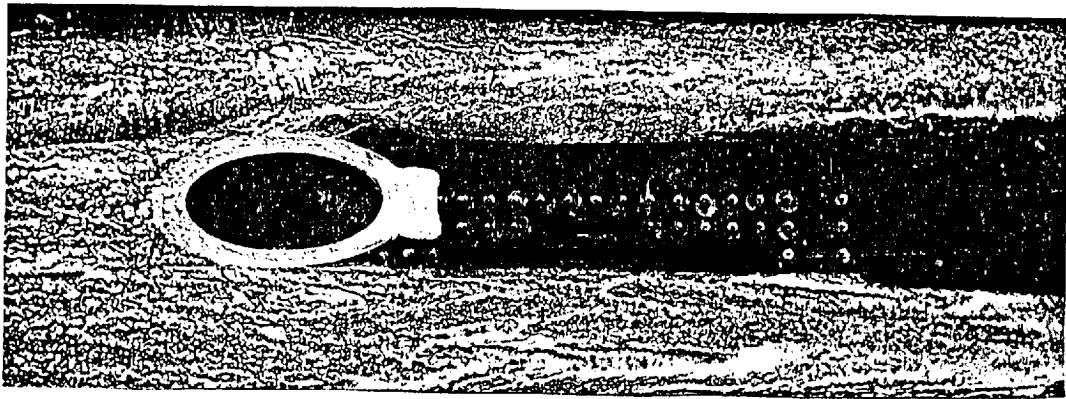


Fig. 28a : Surface flow pattern, $\alpha=30^\circ$, $M=0.20$.

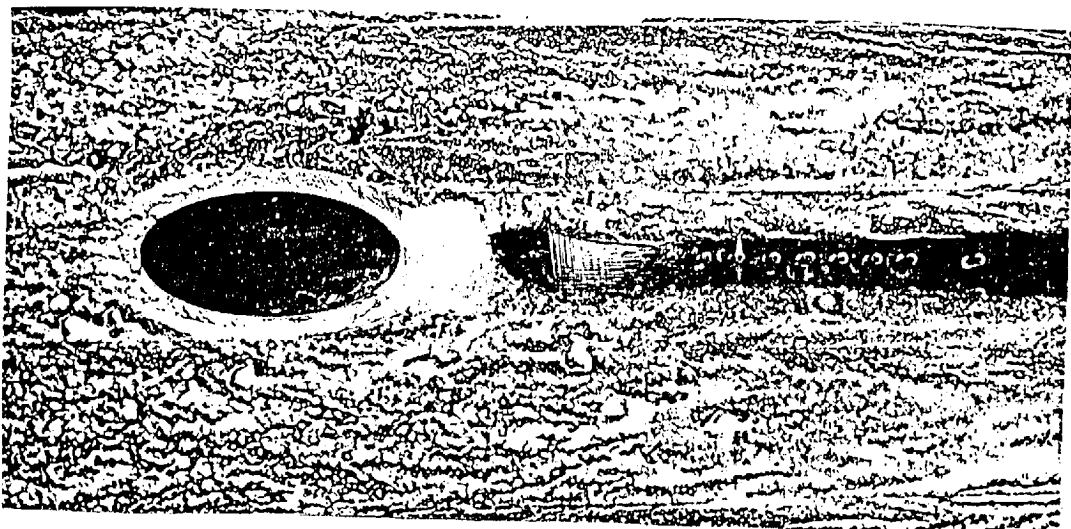


Fig. 28b : Surface flow pattern, $\alpha=30^\circ$, $M=0.30$.

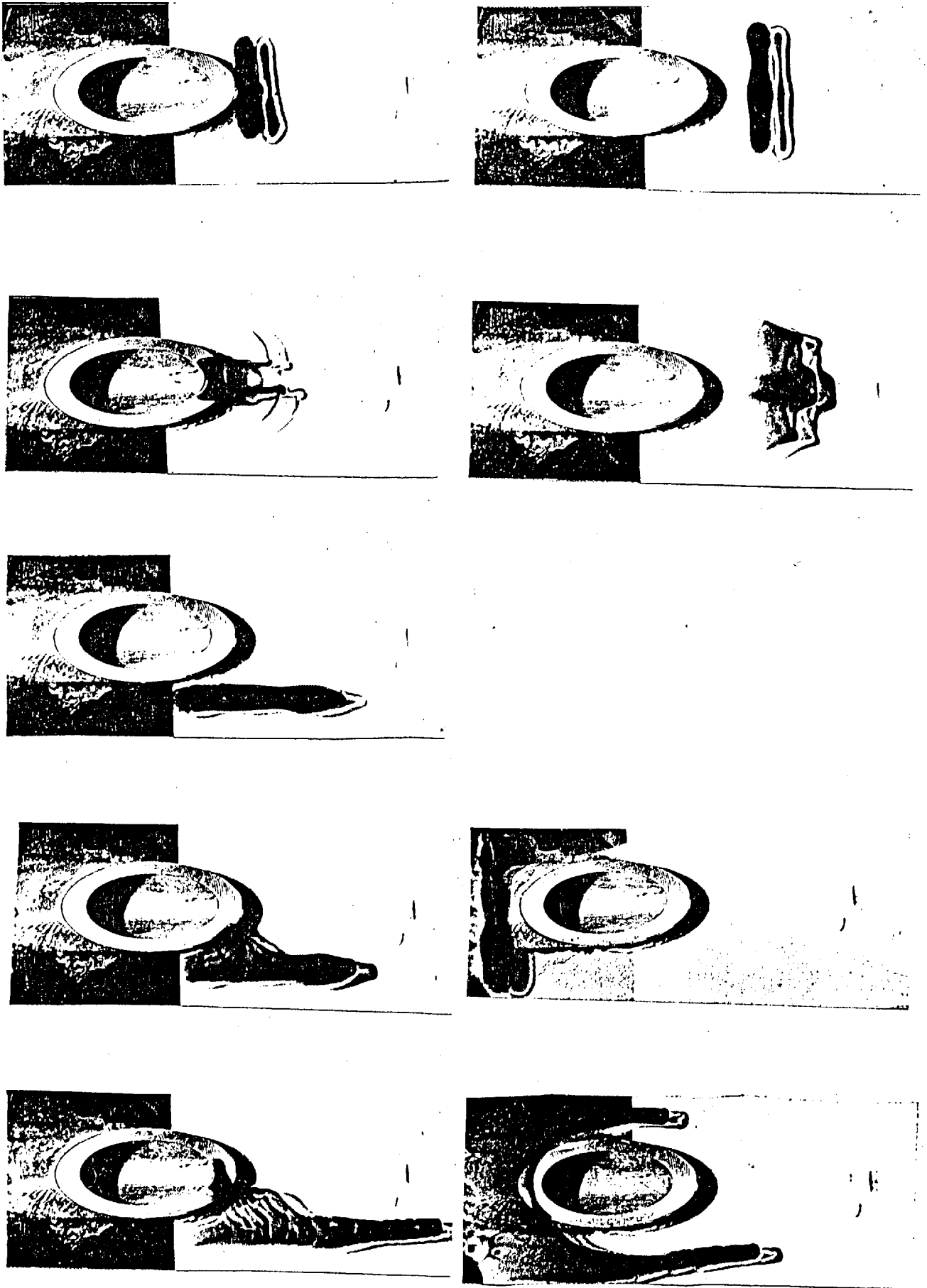
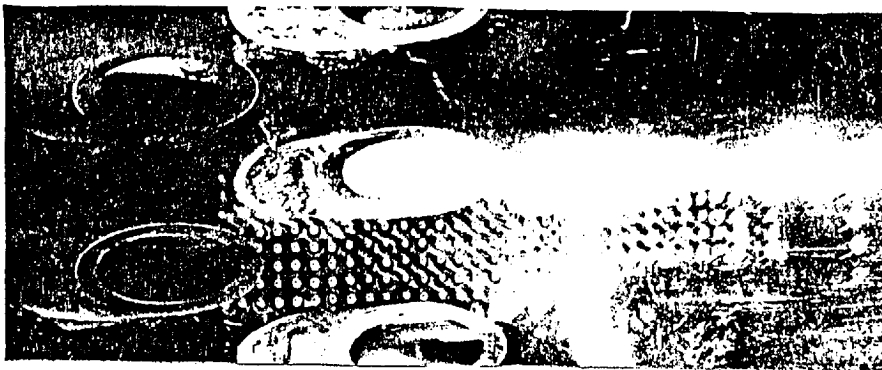


Fig. 28c : Surface flow pattern, $\alpha=30^\circ$, $M=1.0$ (flow from left)



(a) : $M=0.2$



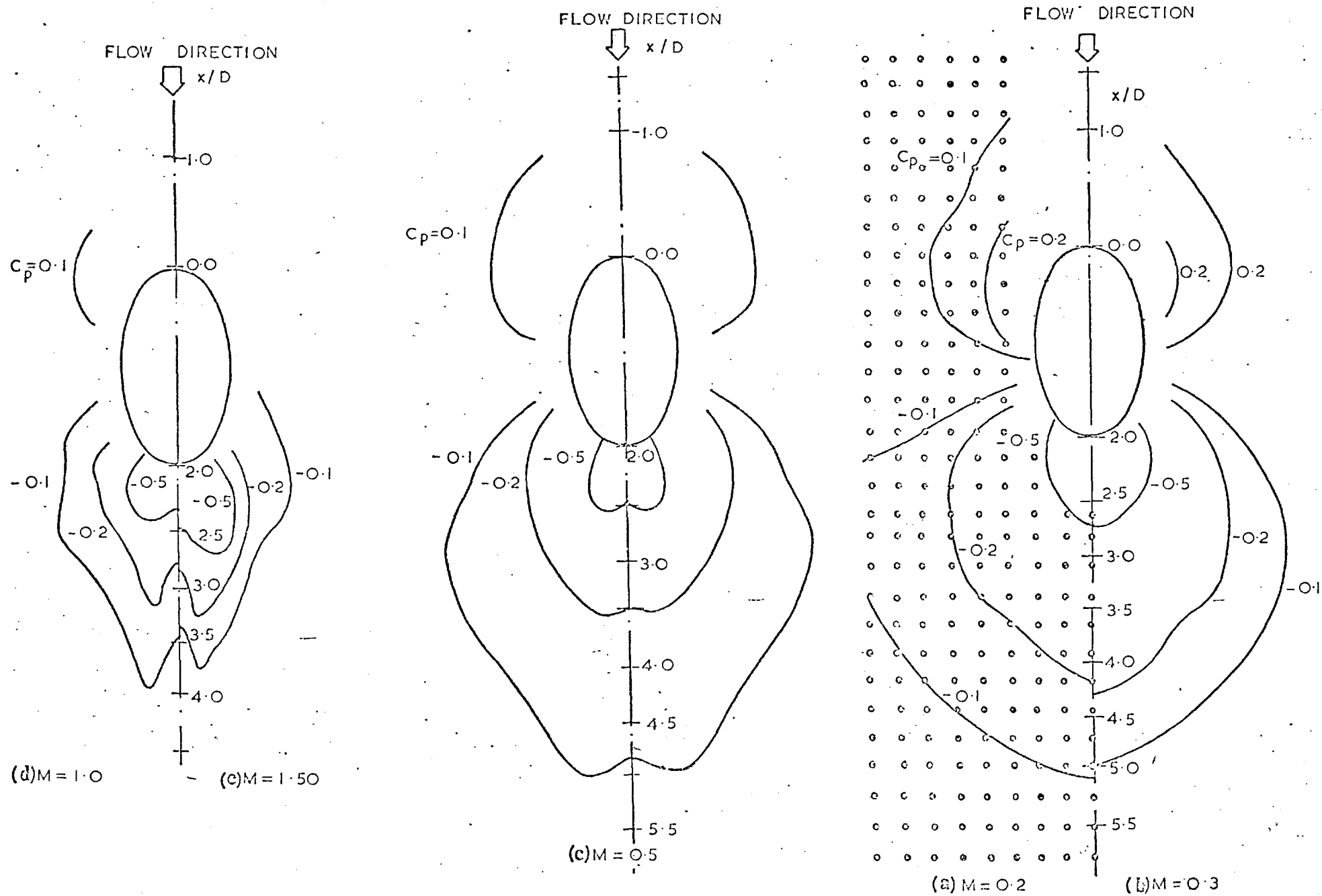
(b) : $M=0.5$



(c) : $M=1.0$

Fig. 29 : Plan view of the jet

Fig. 30 : Surface isobaric contours.



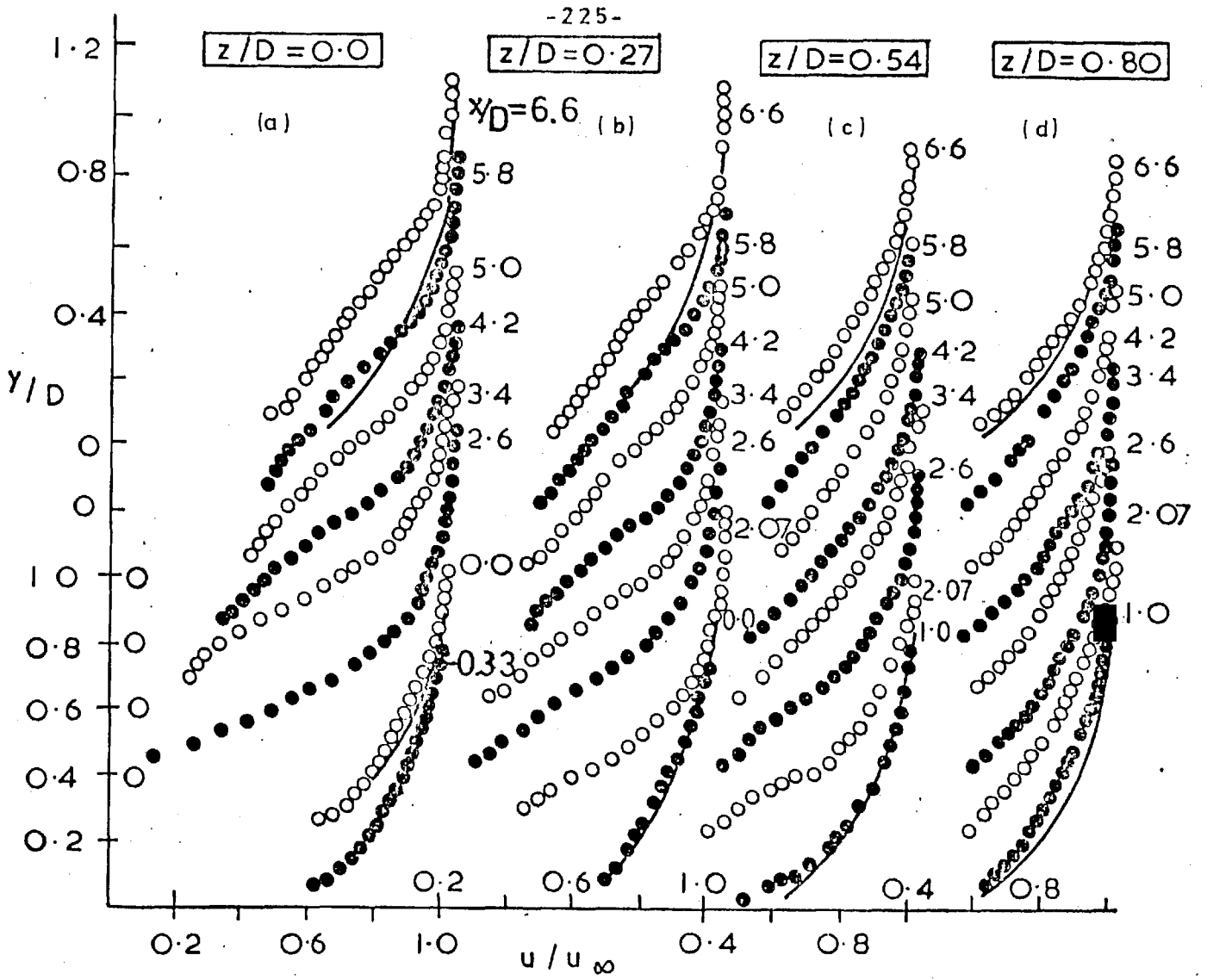


Fig. 31 : Profiles of streamwise velocity

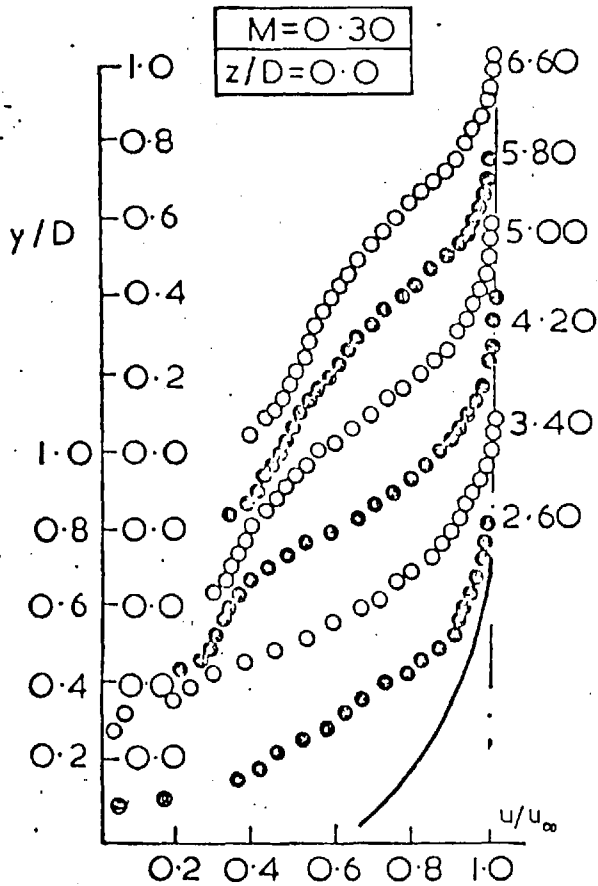


Fig. 32 : Profiles of streamwise velocity

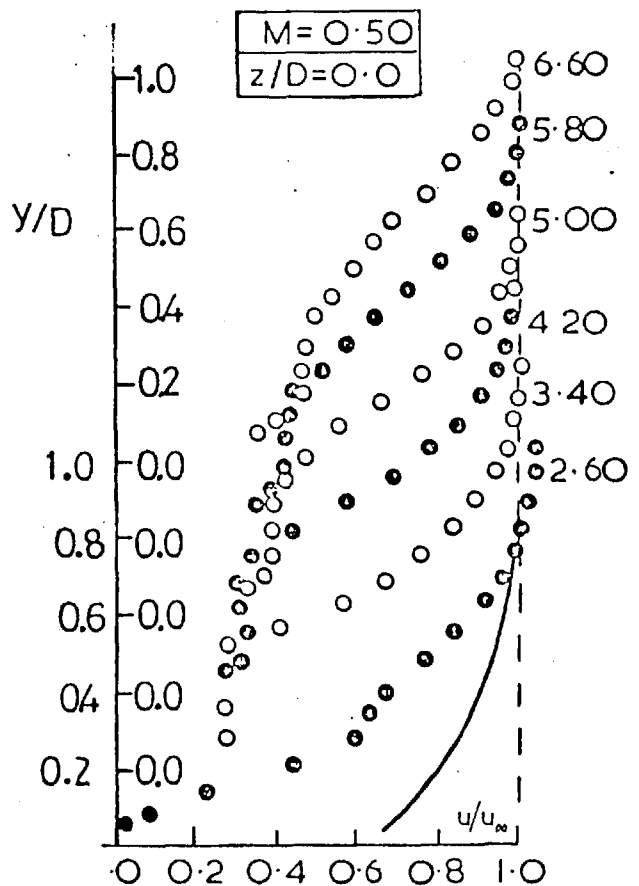


Fig. 33 : Profiles of streamwise velocity

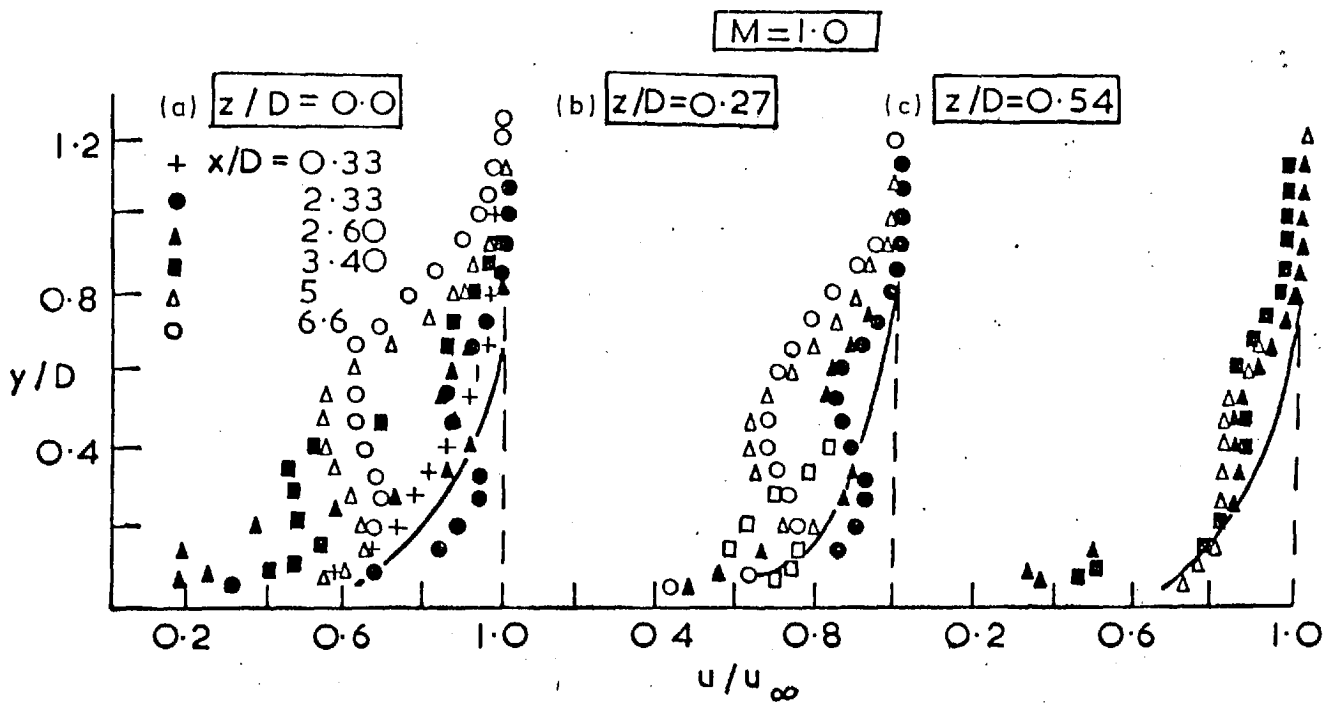


Fig. 34 : Profiles of streamwise velocity,

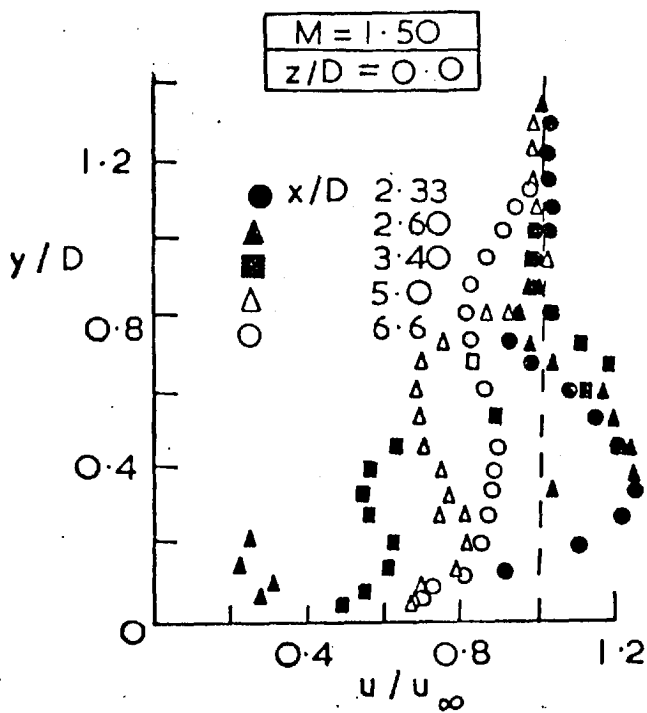


Fig. 35 : Profiles of streamwise velocity

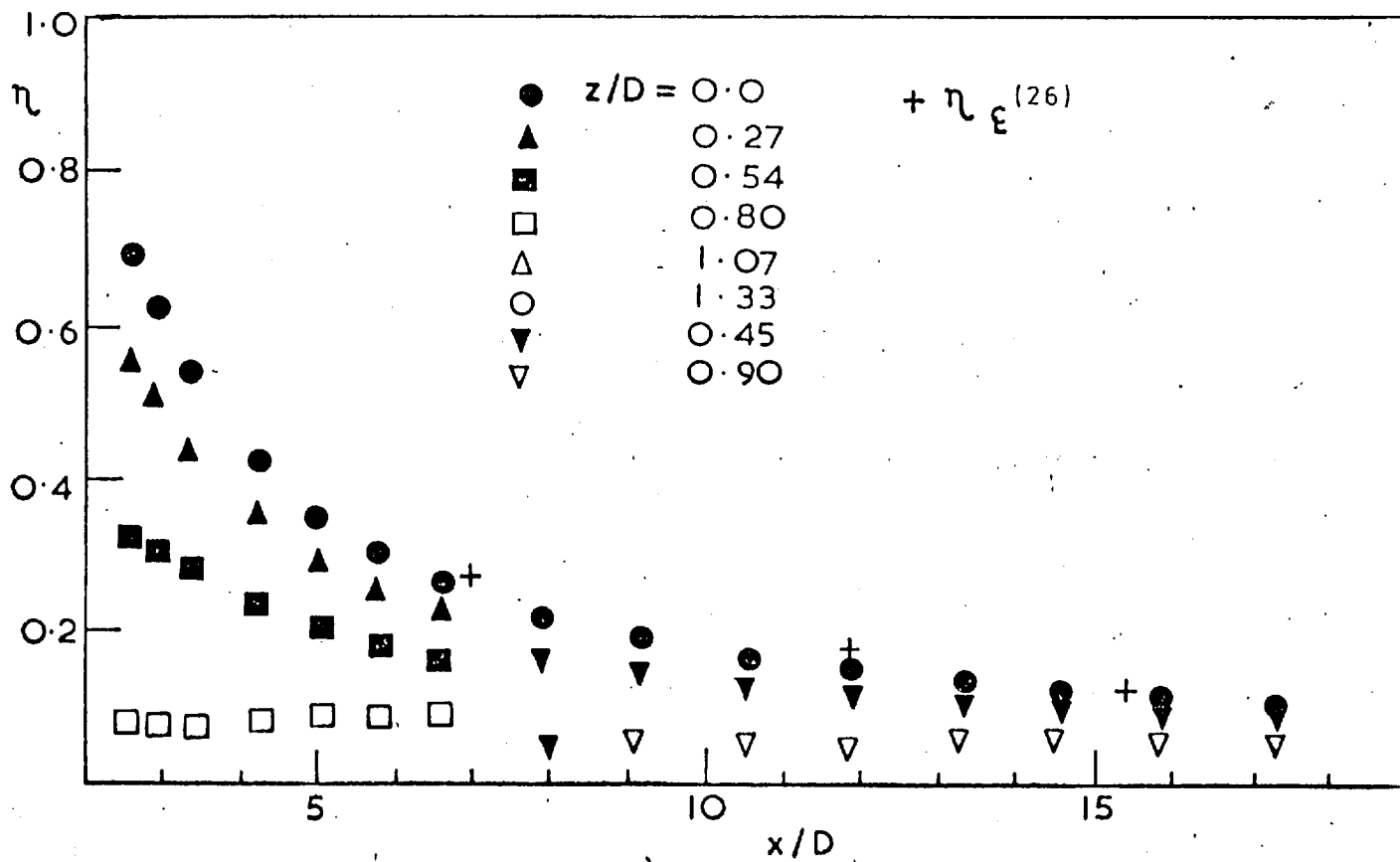


Fig.36a: Wall film cooling effectiveness, $M=0.1$

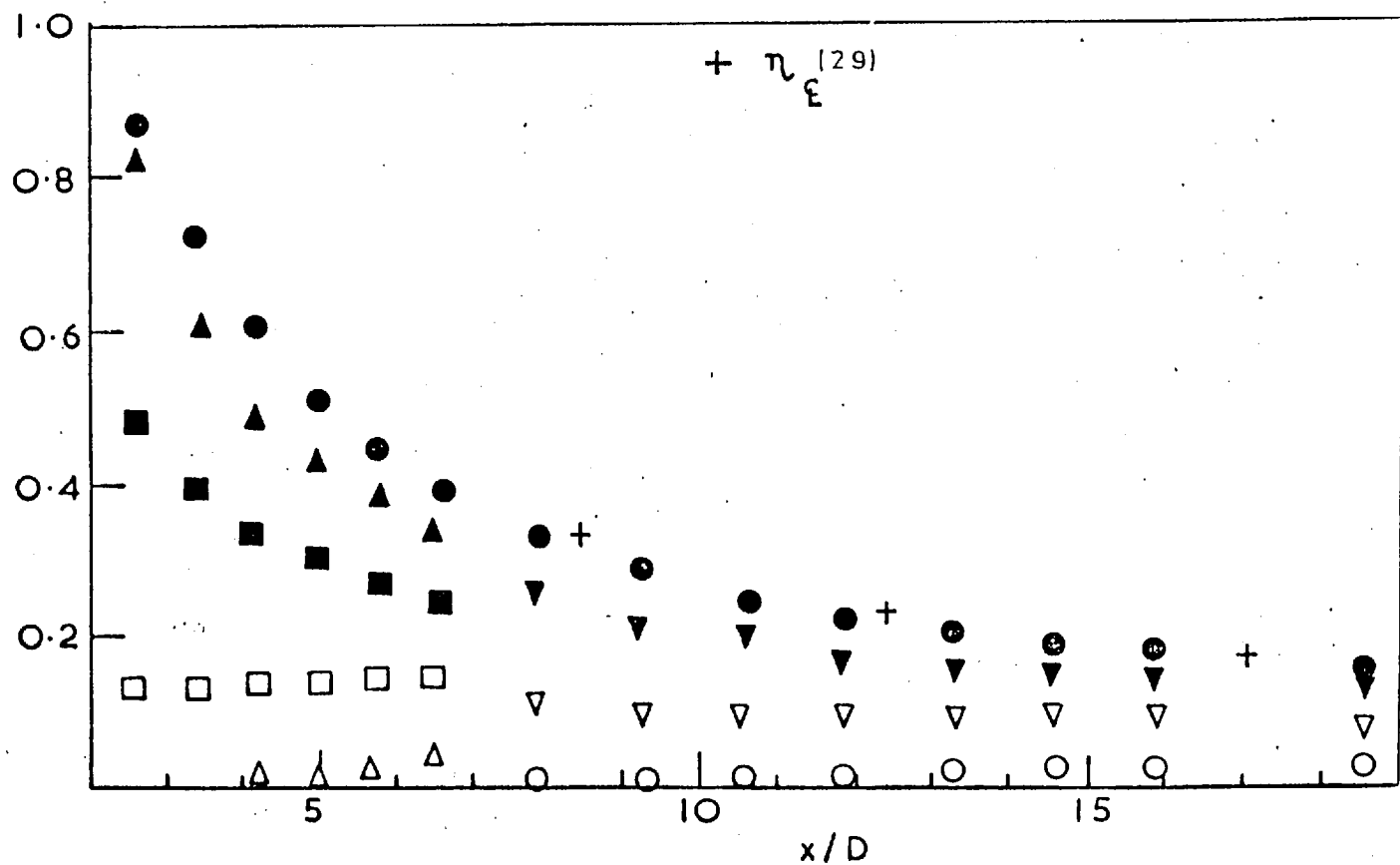


Fig.36b: Wall film cooling effectiveness, $M=0.2$

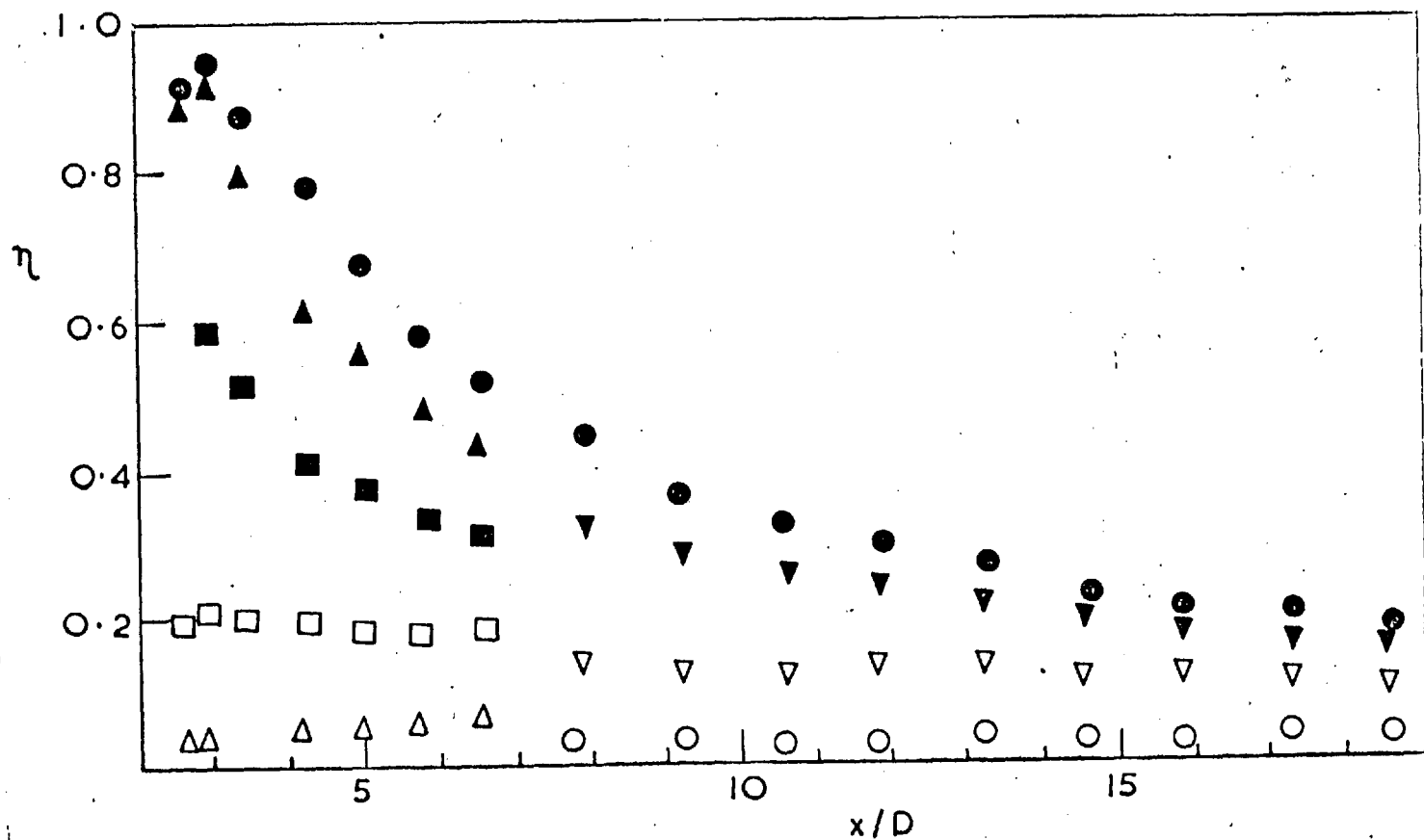


Fig.36c: Wall film cooling effectiveness, $M=0.3$

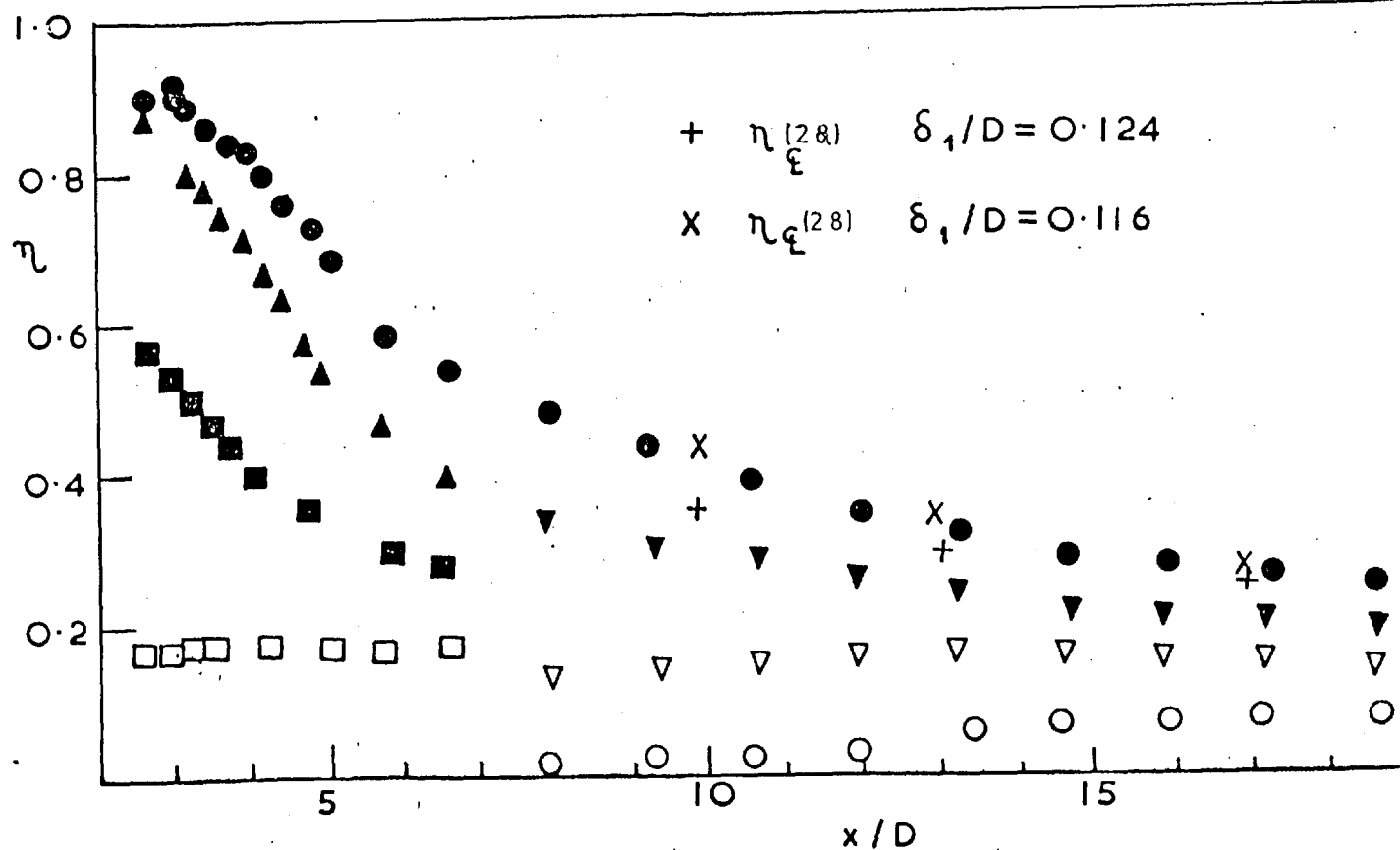


Fig.36d: Wall film cooling effectiveness, $M=0.5$

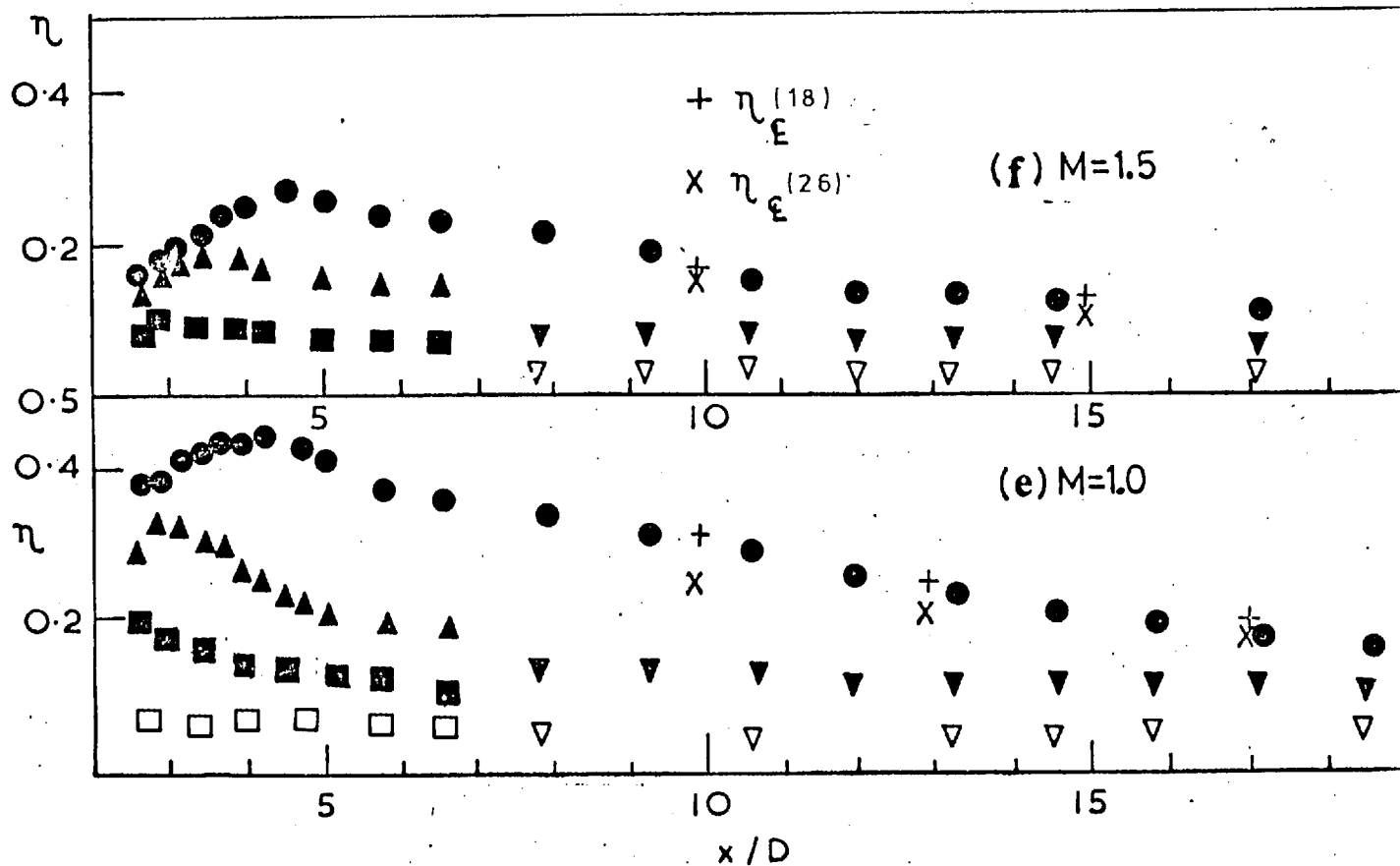


Fig. 36 : Wall film cooling effectiveness.

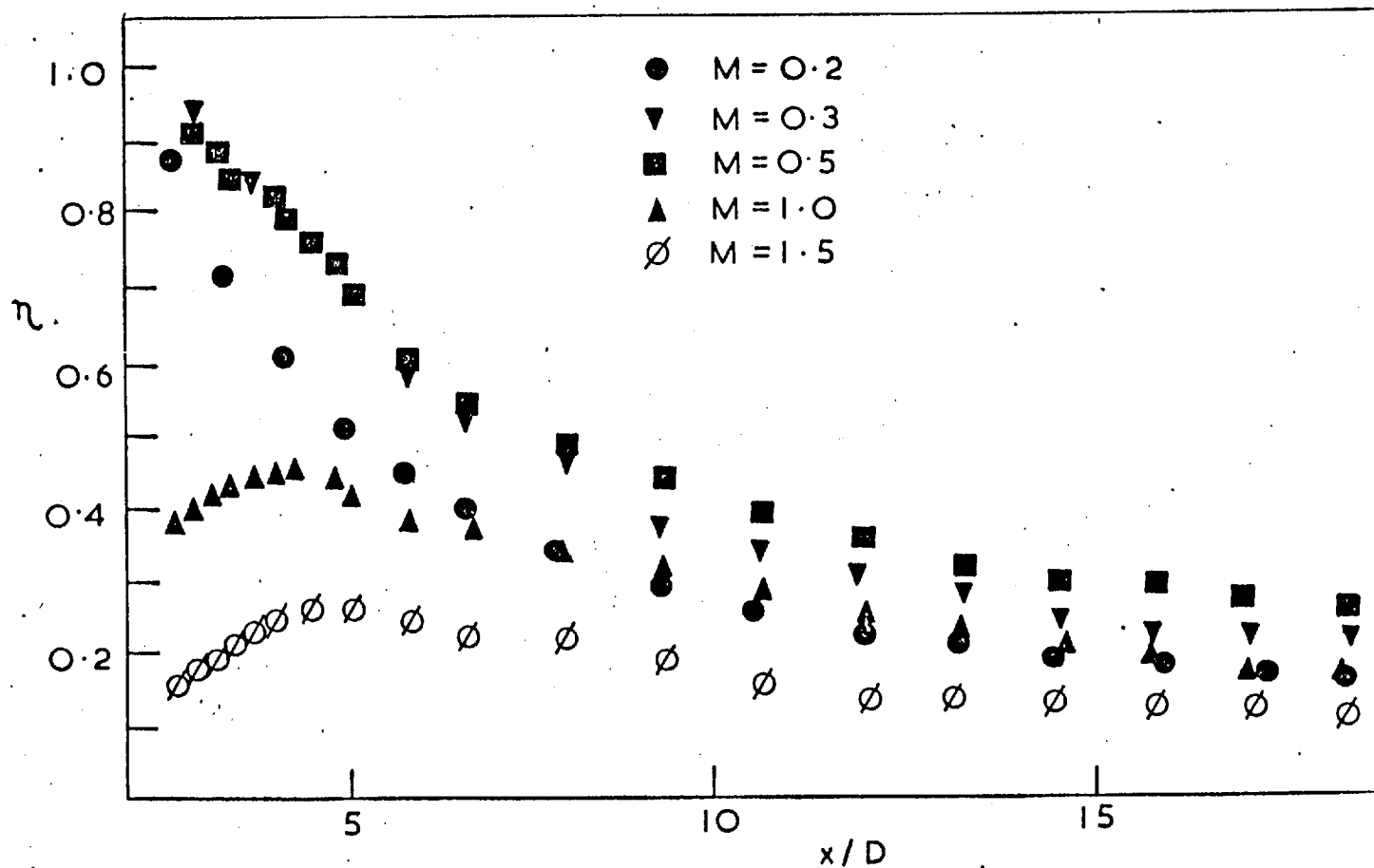


Fig. 37a : Centre-line effectiveness, effect of M .

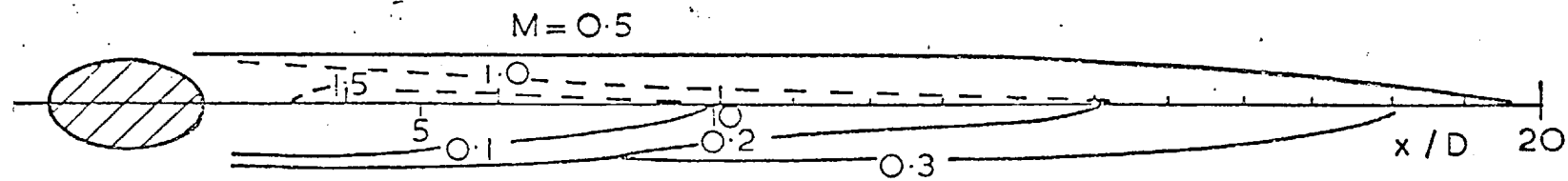


Fig. 37b : Lateral spread of the jet, effect of M .



Fig. 38 : Plan view of jets, $M = 0.5$.

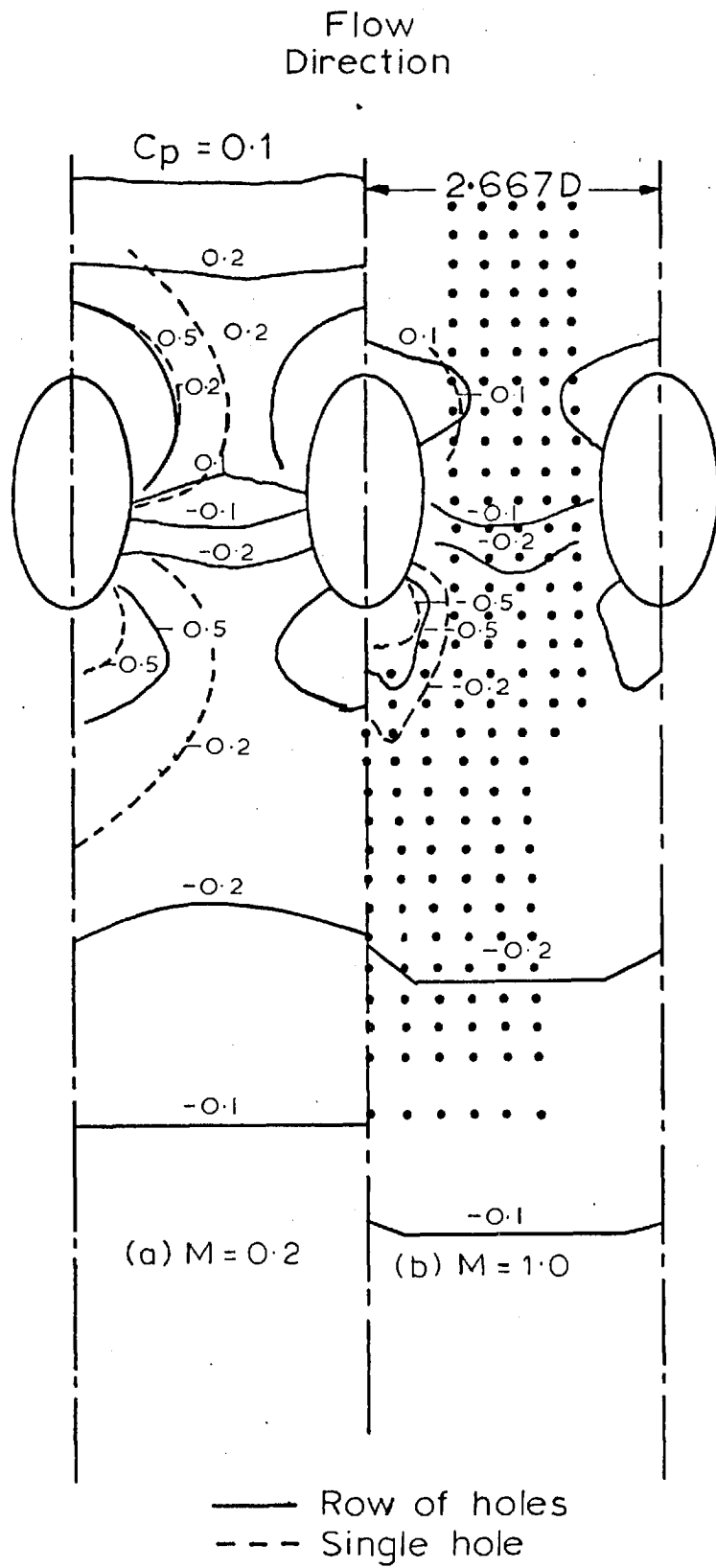


Fig. 39 : Surface isobaric contours.

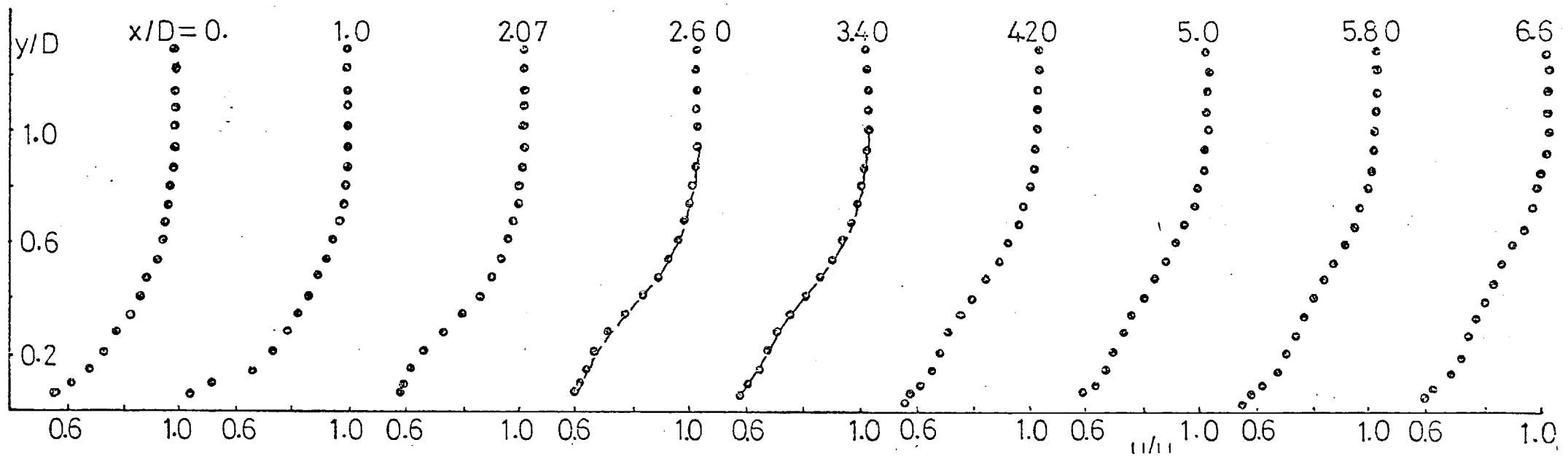


Fig. 40b : Streamwise velocity profiles, $z/D=0.53$, $M=0.5$.

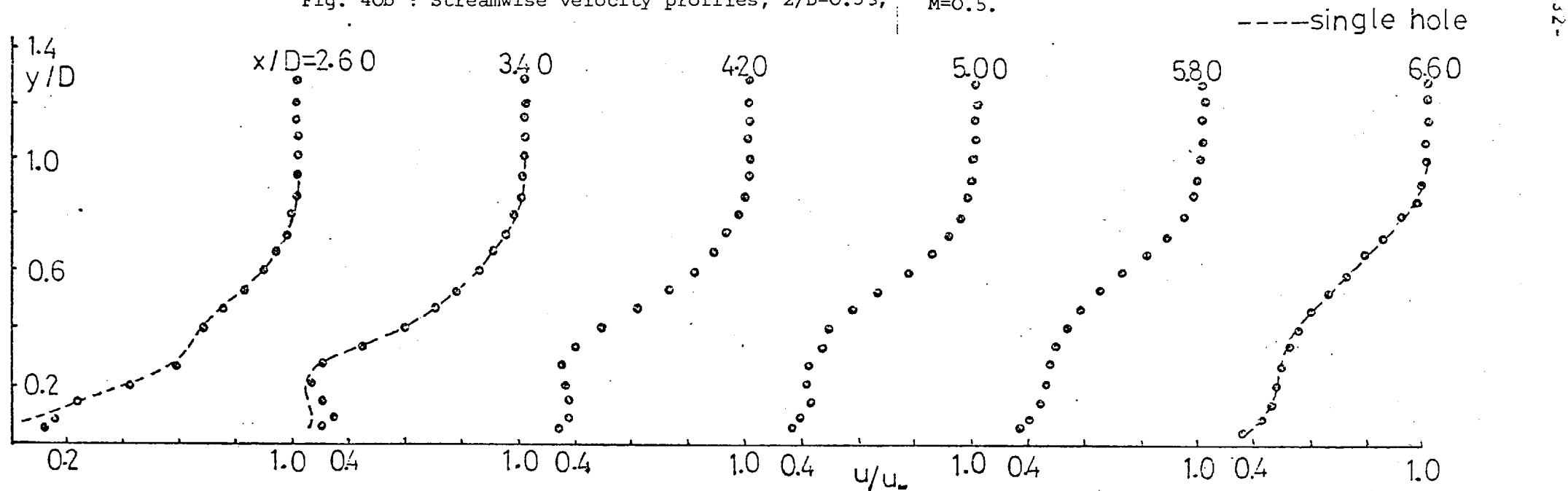


Fig. 40a : Streamwise velocity profiles, $z/D=0.0$, $M=0.5$.

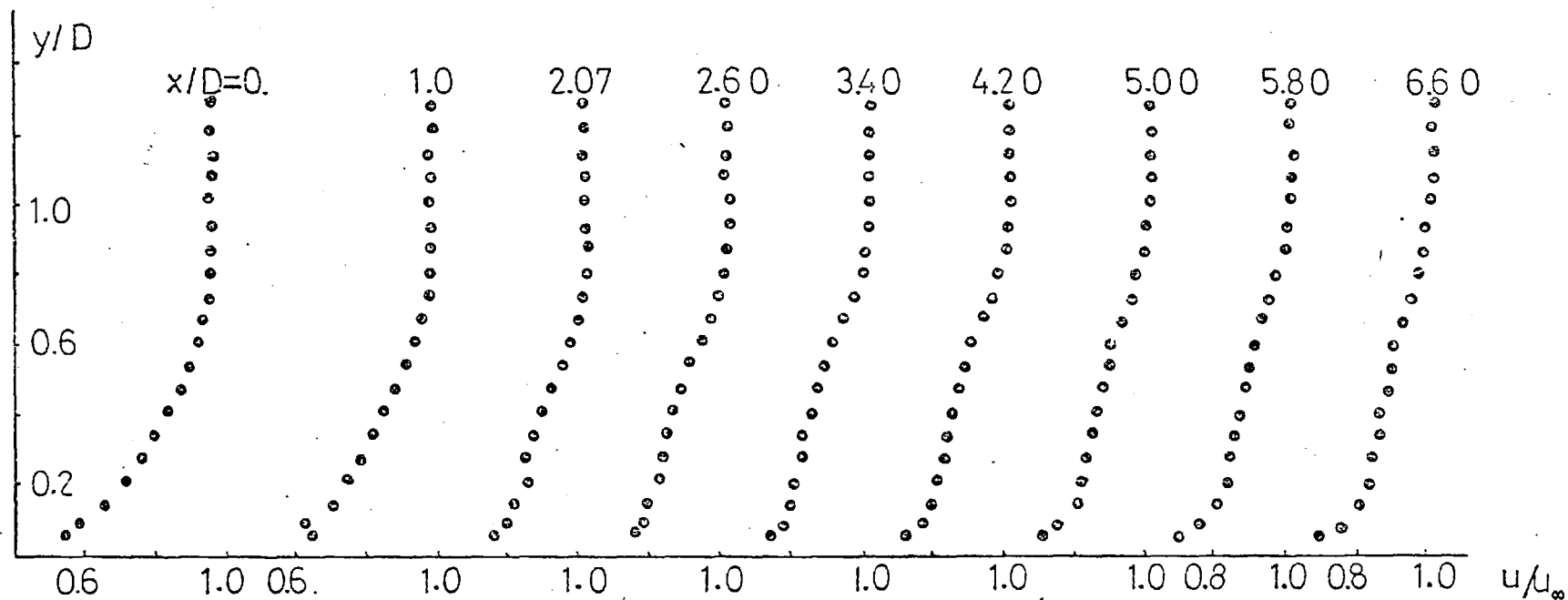


Fig. 40d : Streamwise velocity profiles, $z/D=0.53$, $M=1.0$.

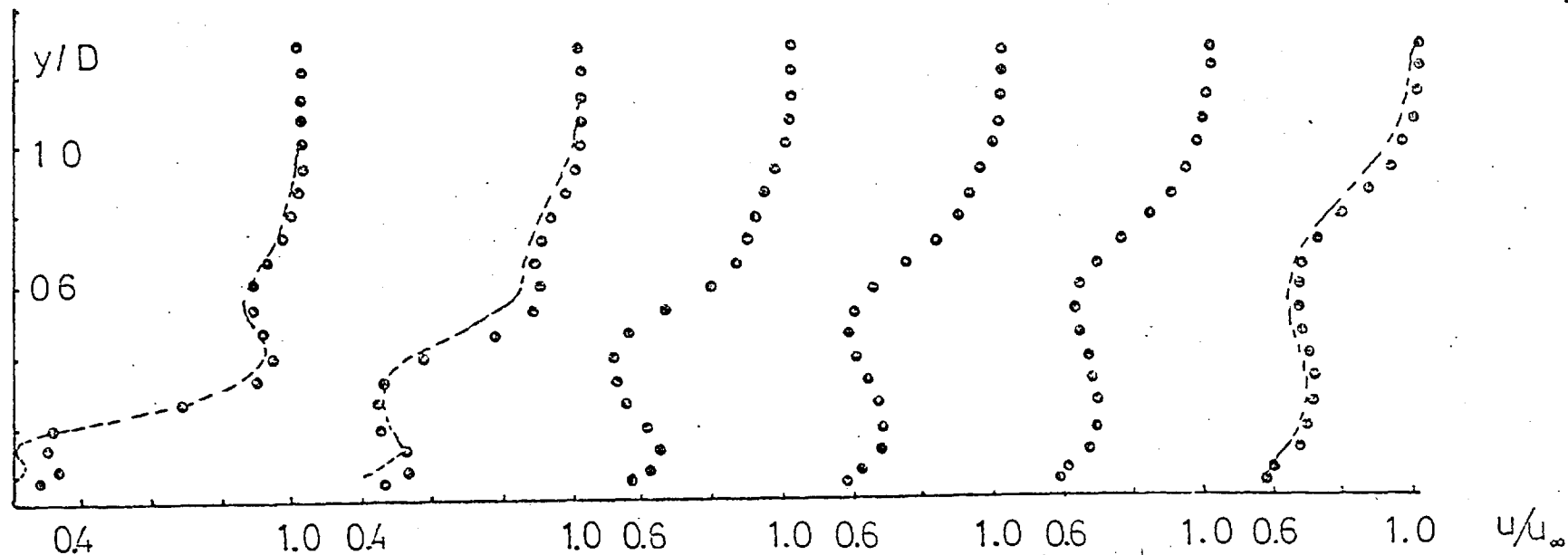


Fig. 40c : Streamwise velocity profiles, $z/D=0.0$, $M=1.0$.

---- single hole

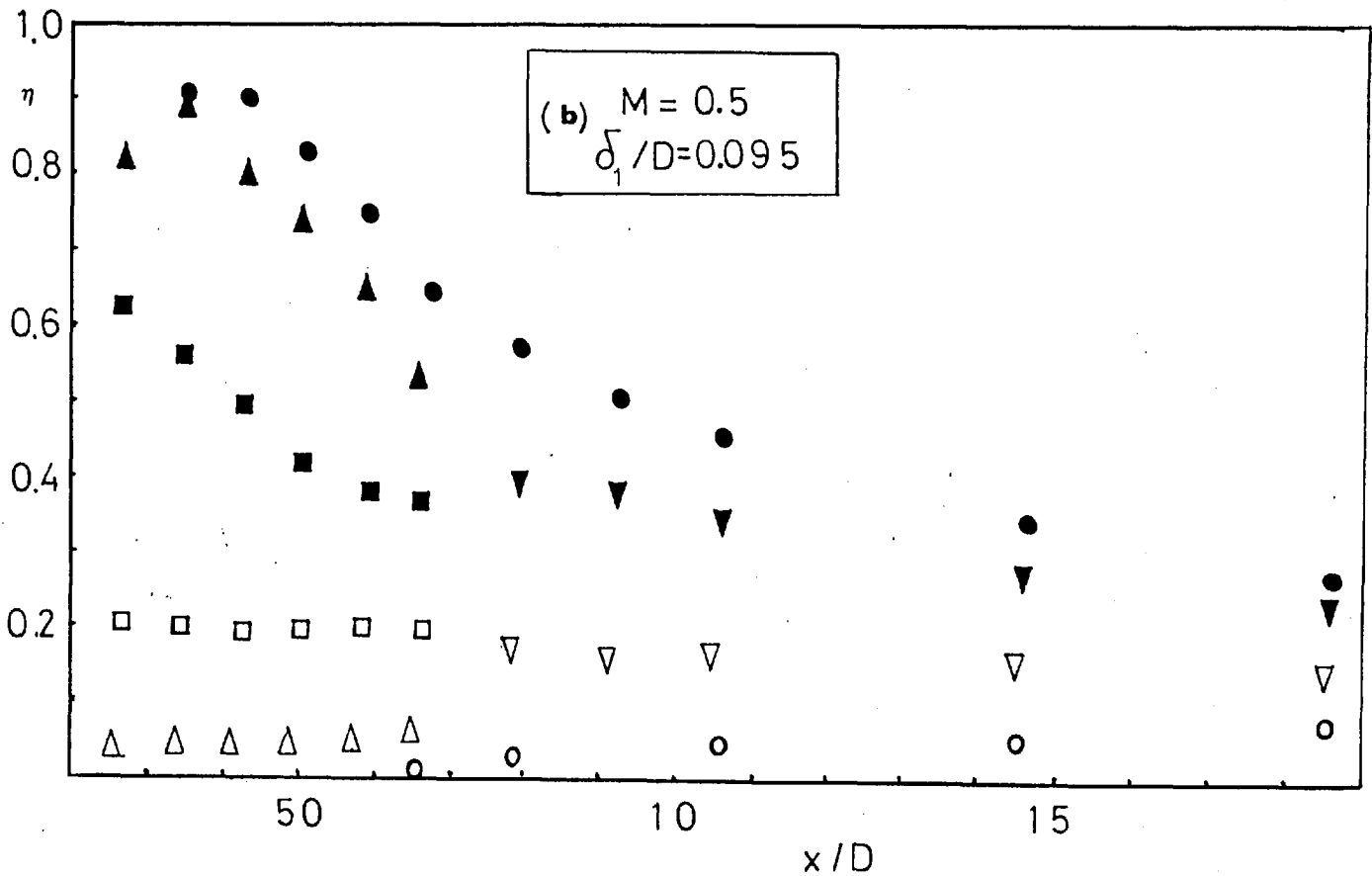
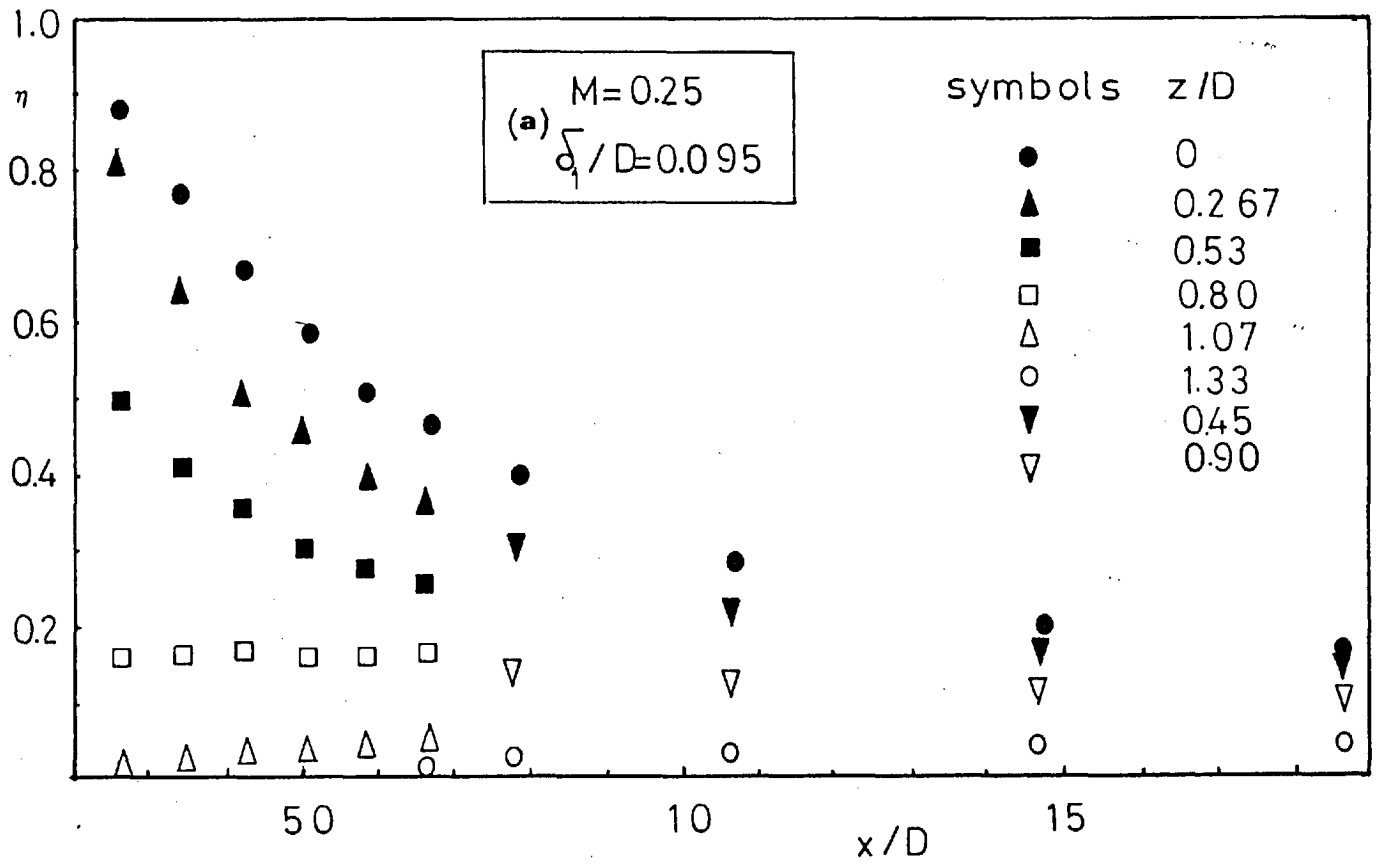


Fig. 41: Wall film cooling effectiveness, Row of holes

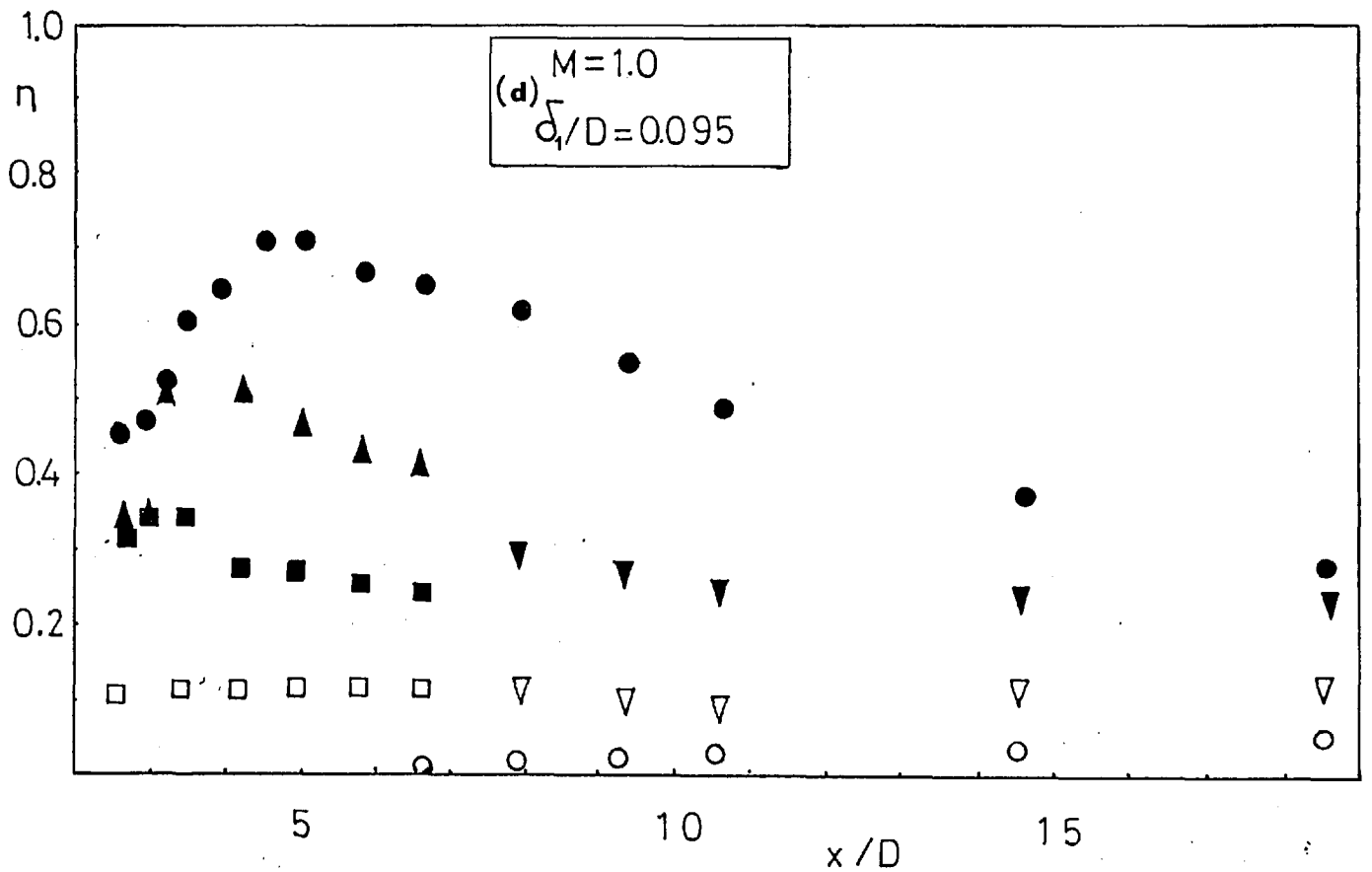
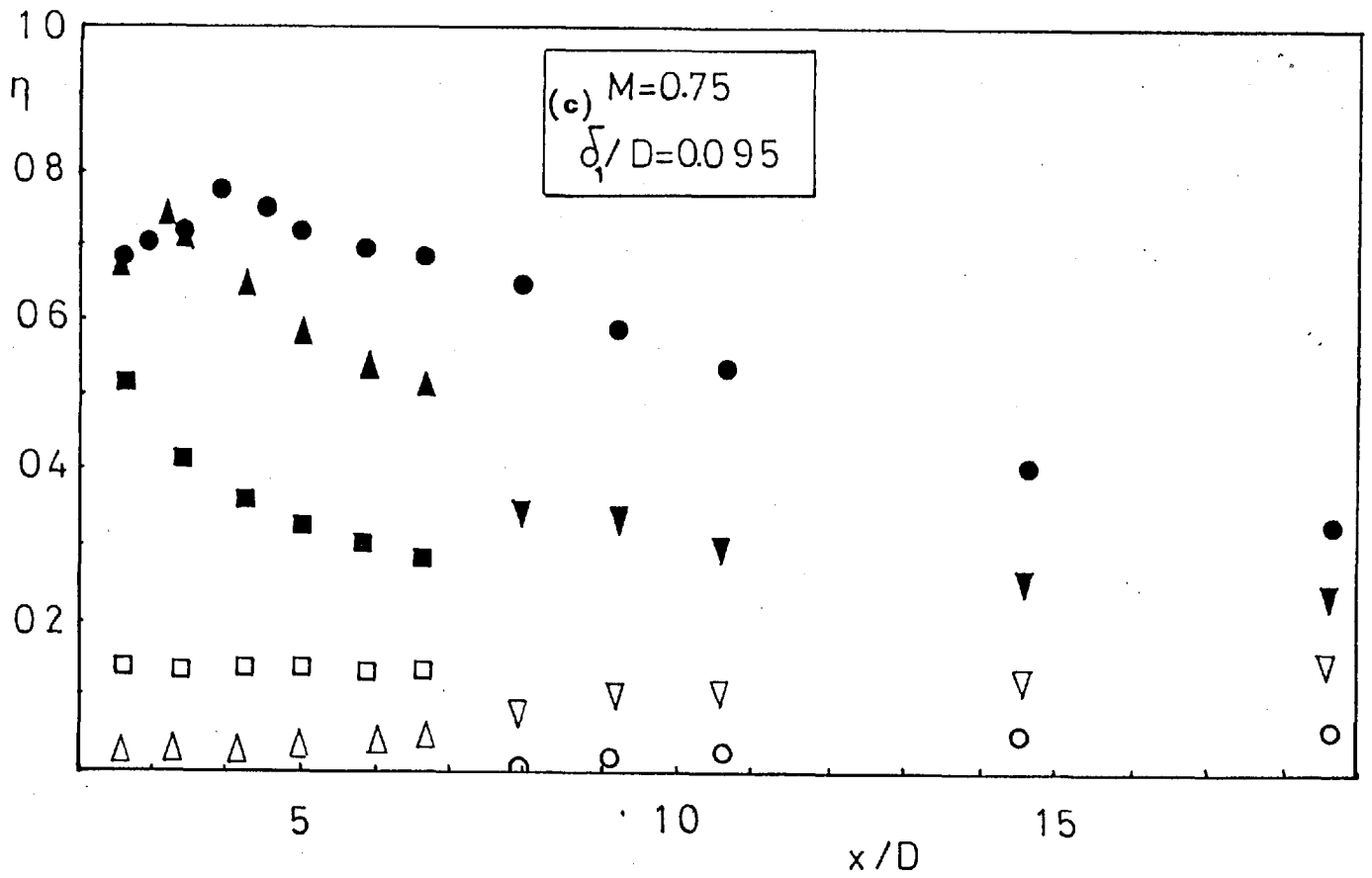


Fig. 41: Wall film cooling effectiveness, Row of holes

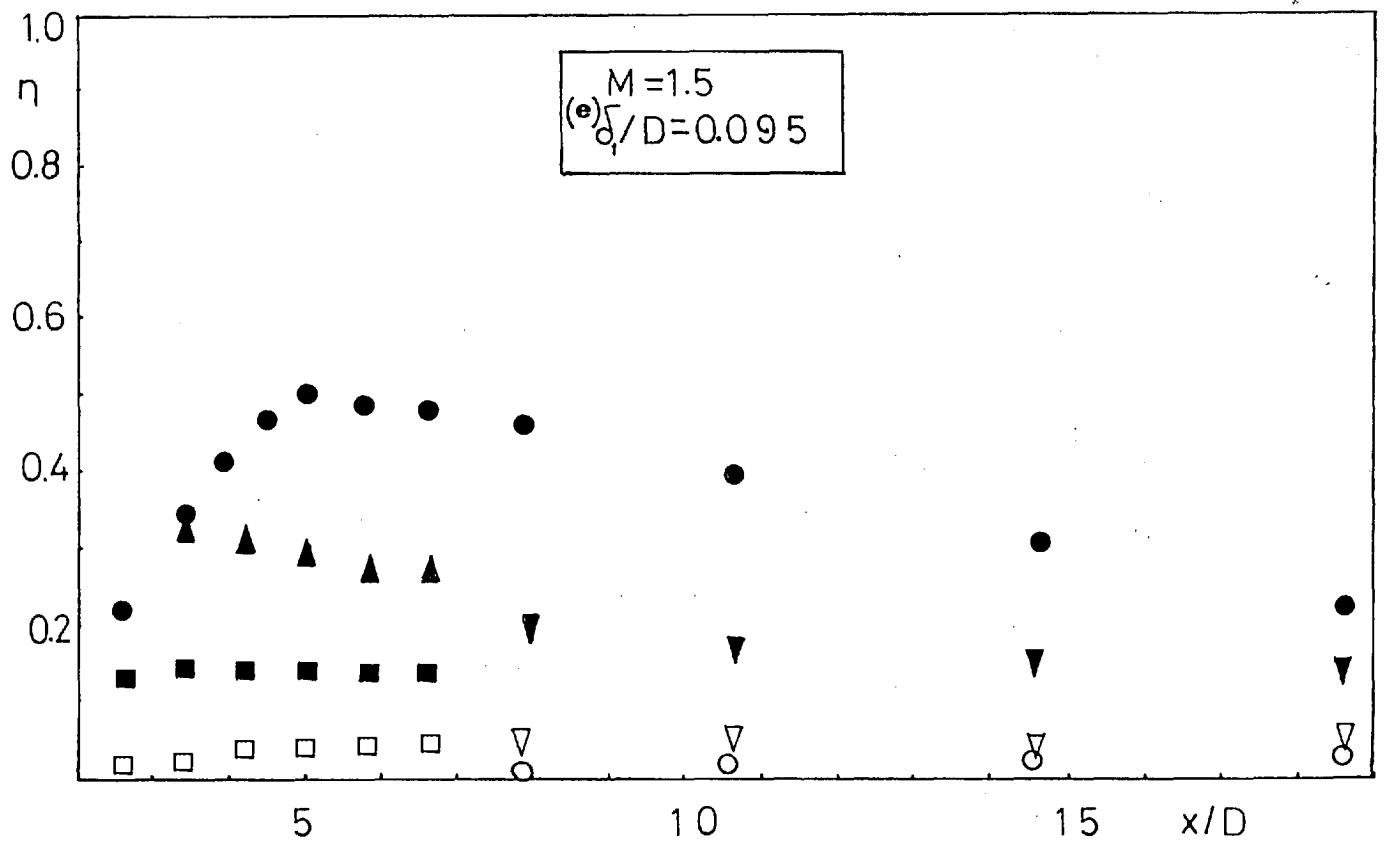


Fig. 41: Wall film cooling effectiveness, Row of holes

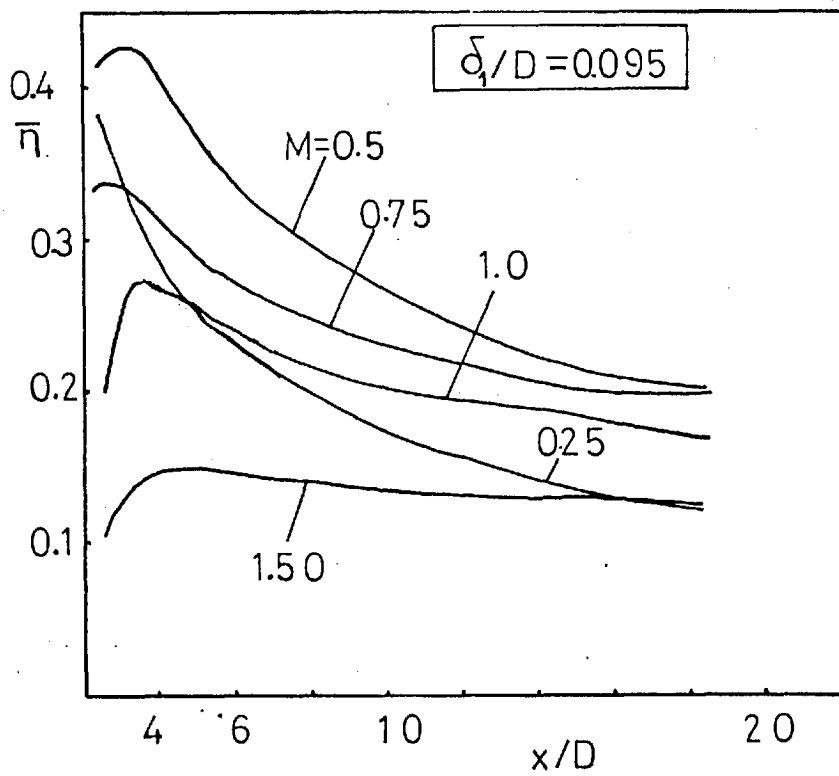


Fig. 42: Laterally averaged wall effectiveness, Row of holes

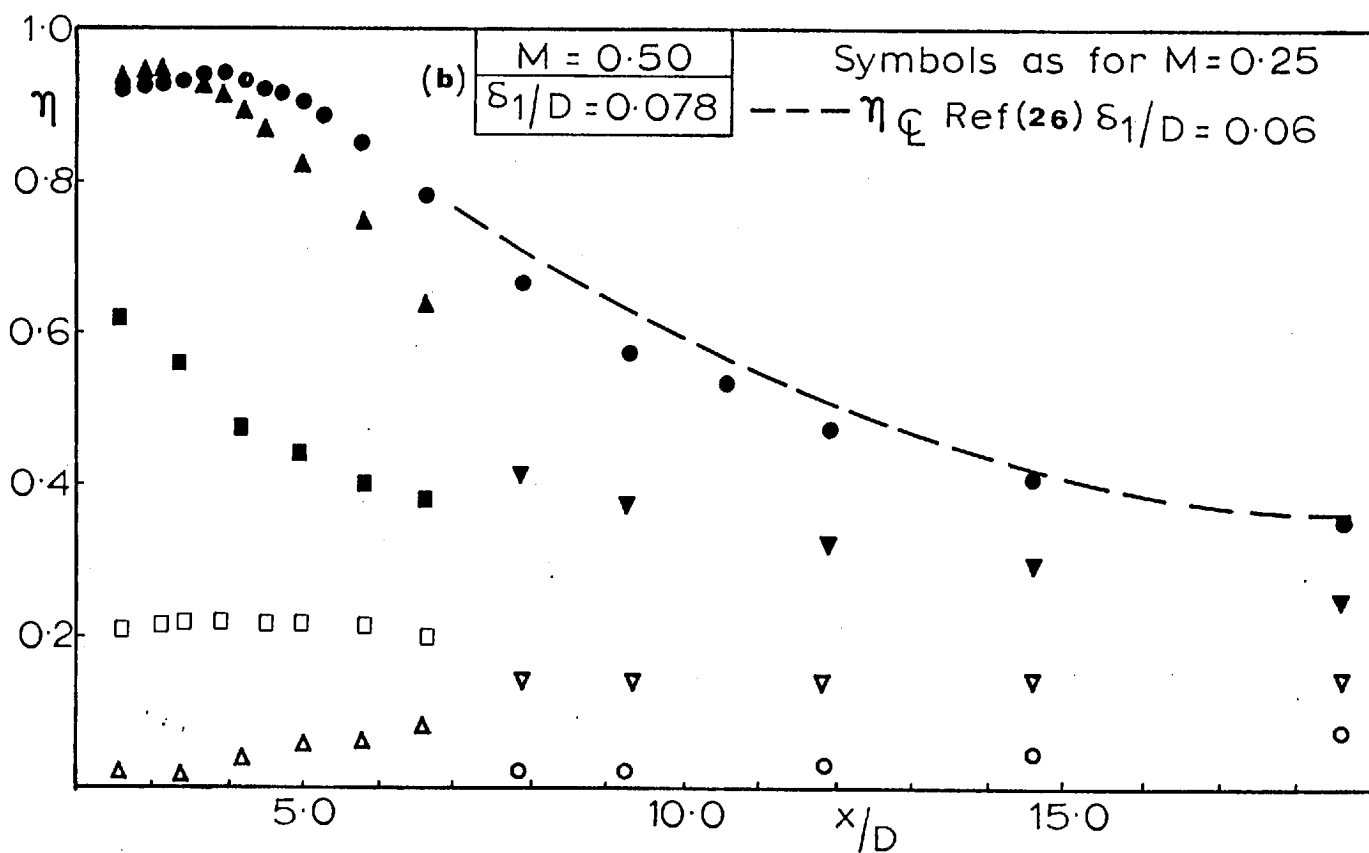
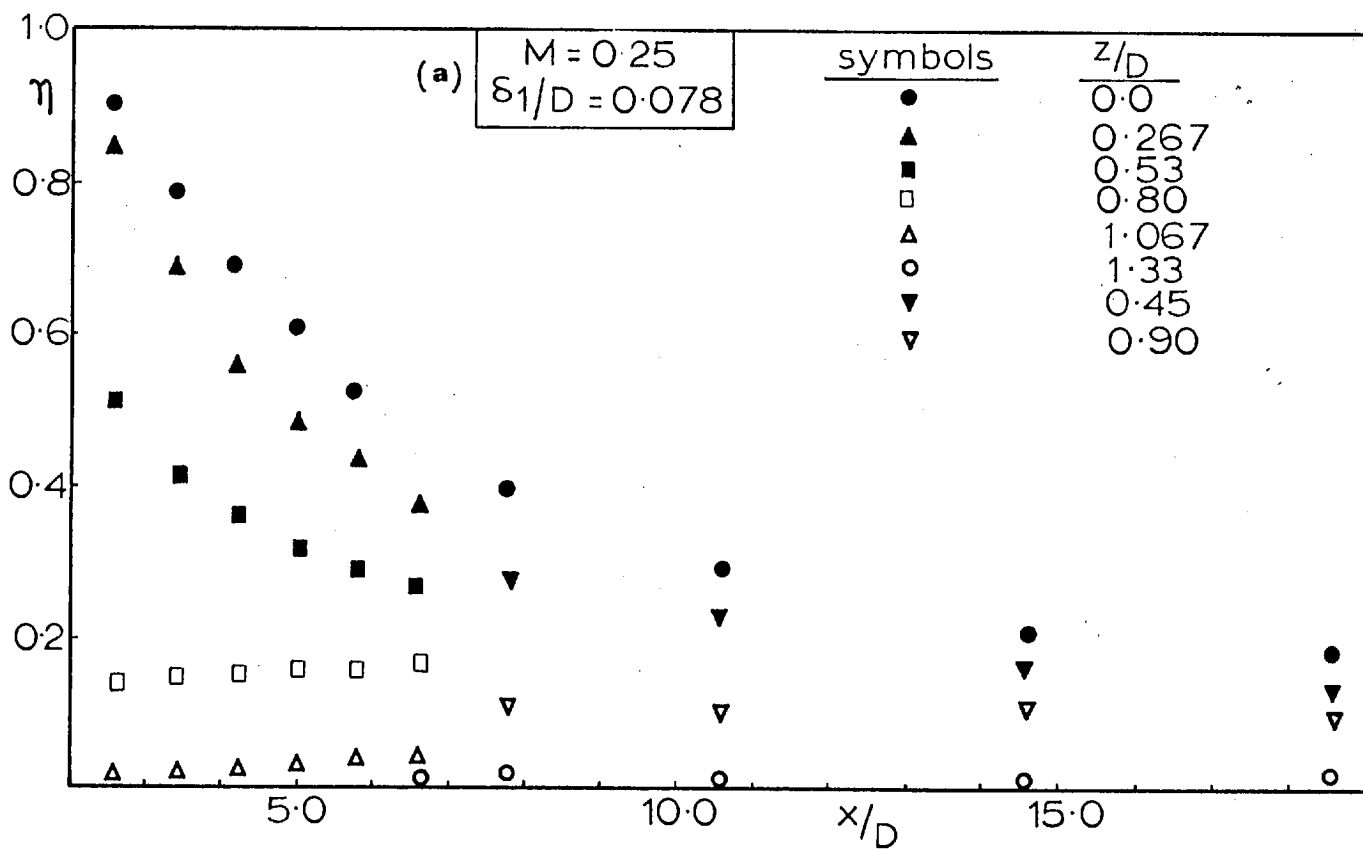


Fig. 43: Wall film cooling effectiveness, Row of holes

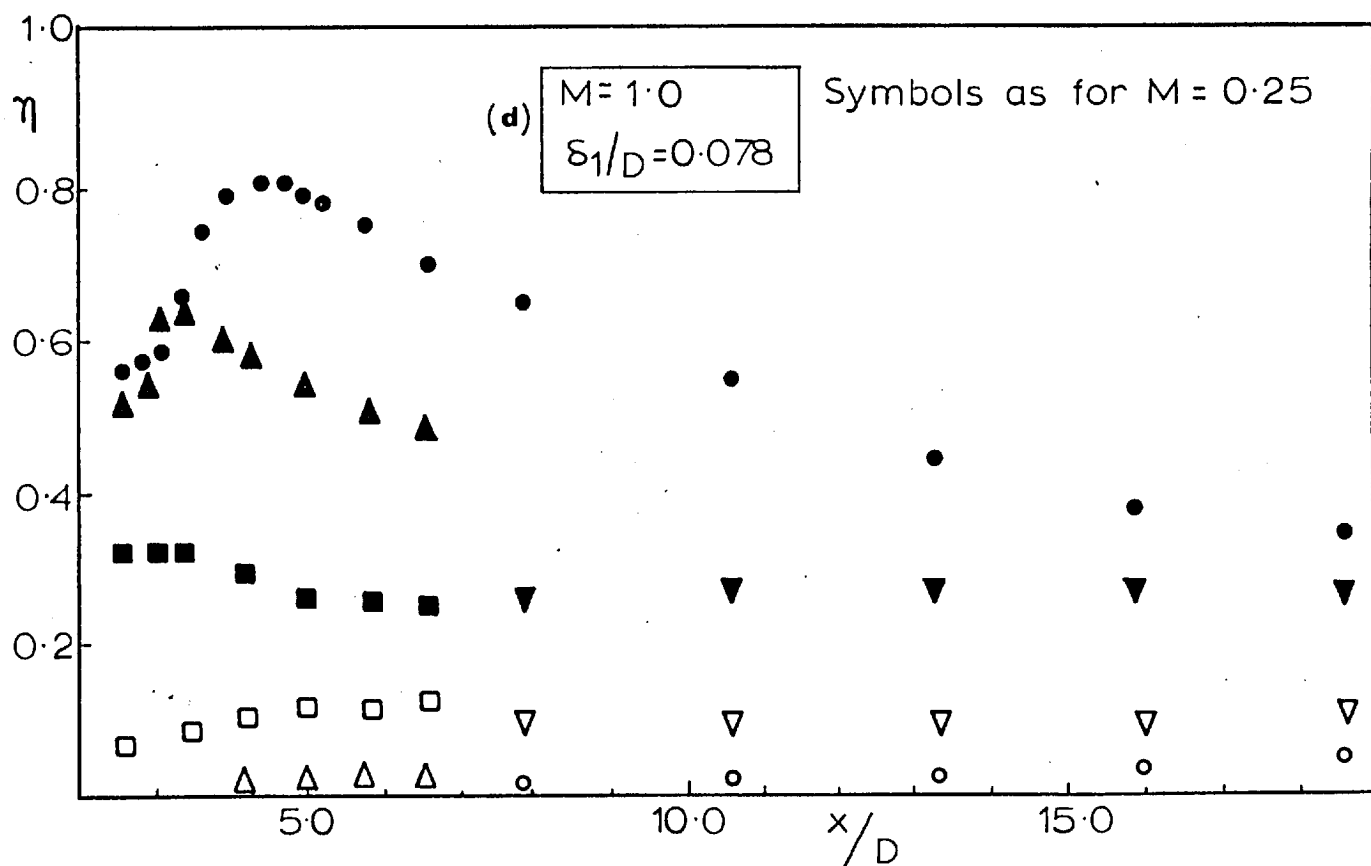
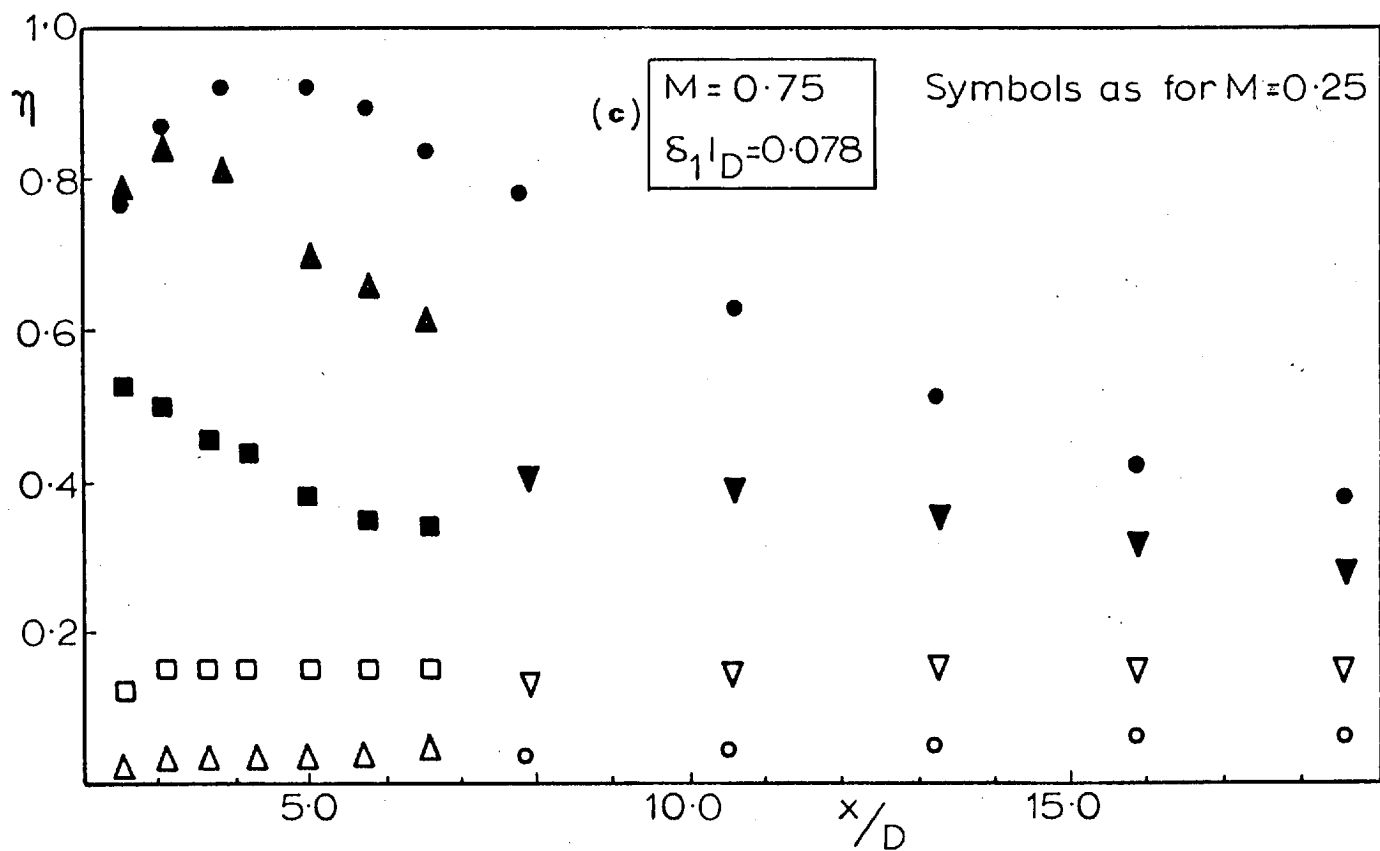


Fig. 43: Wall film cooling effectiveness, Row of holes

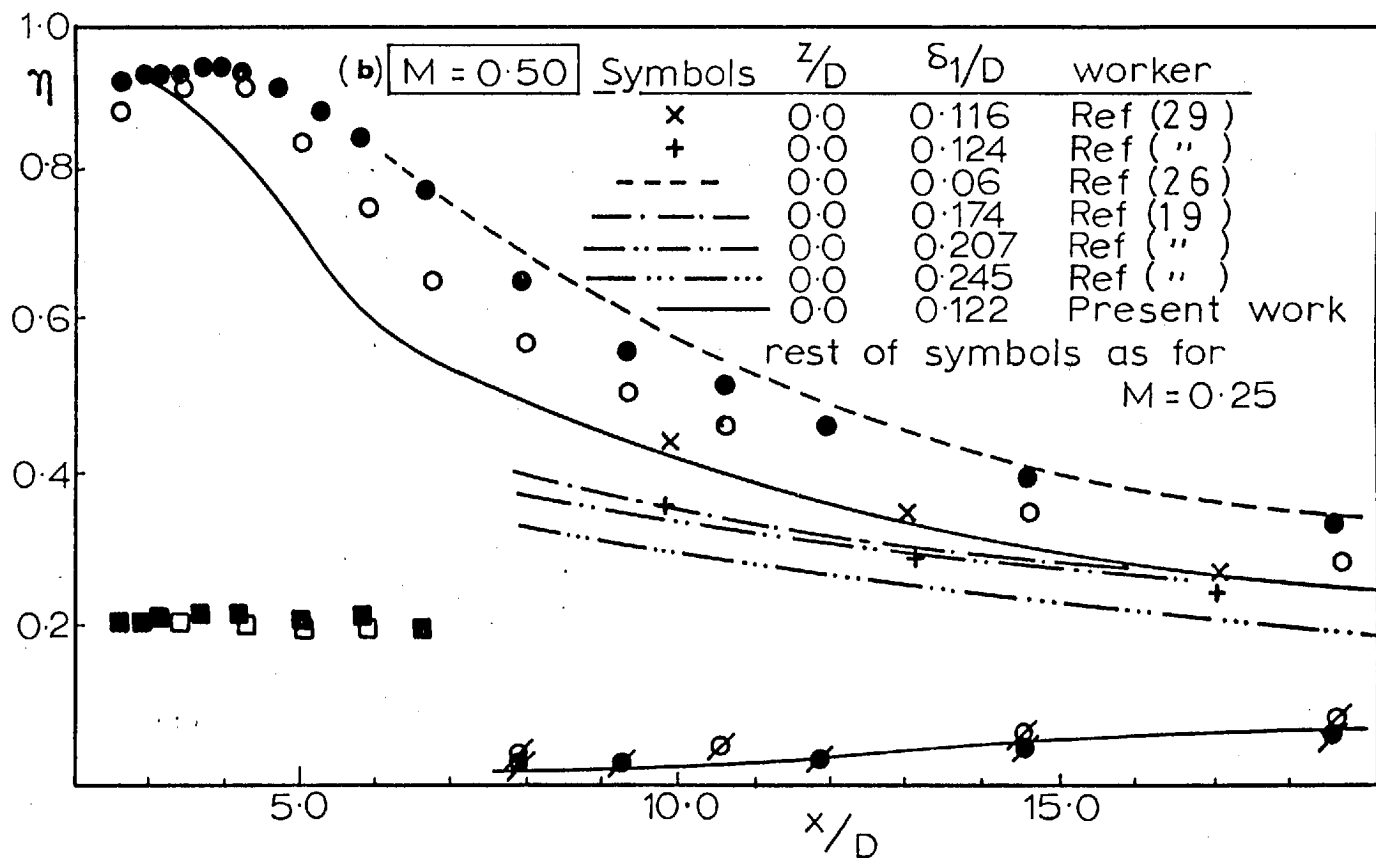
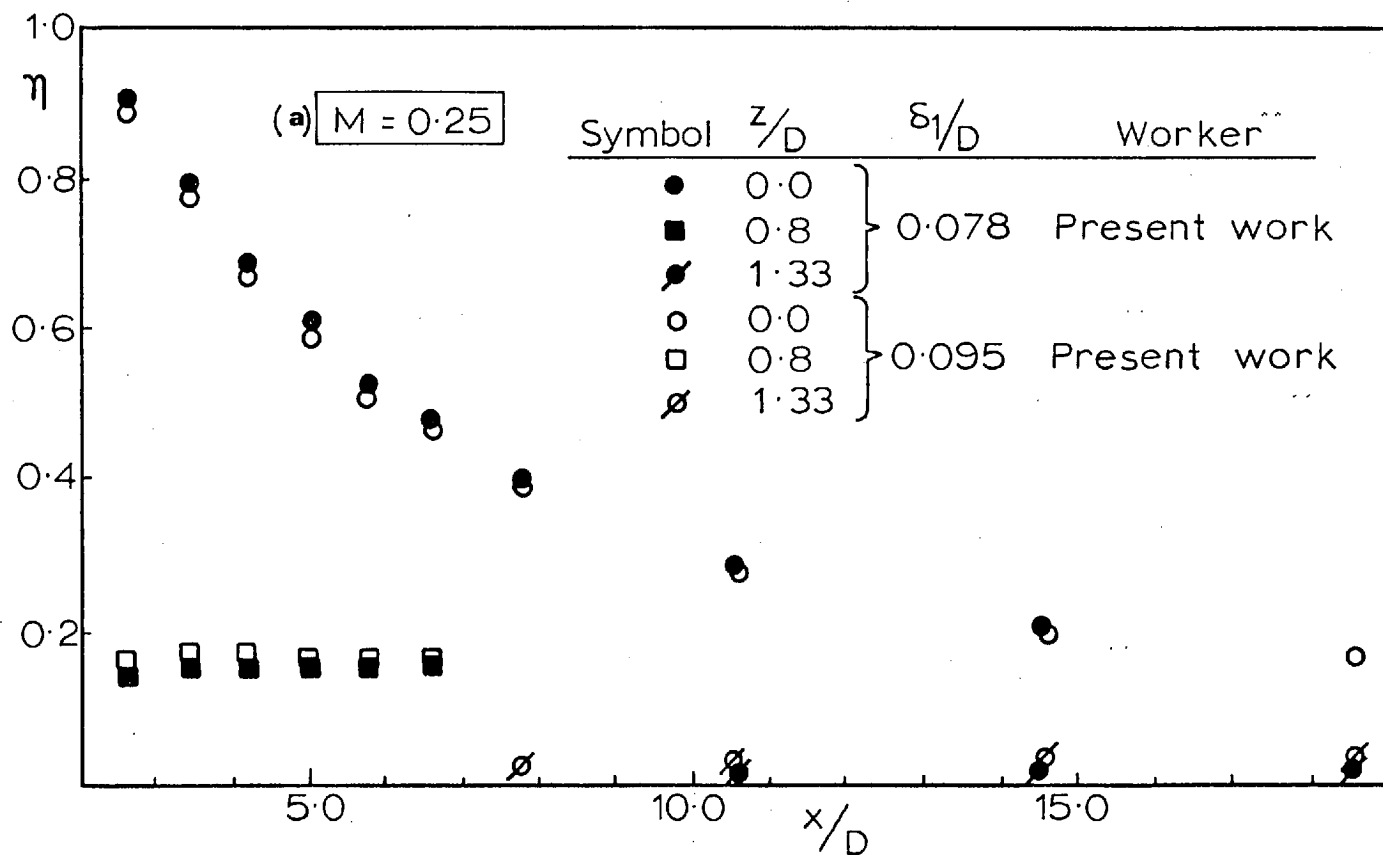


Fig. 44: Effect of boundary layer thickness on η , Row of holes

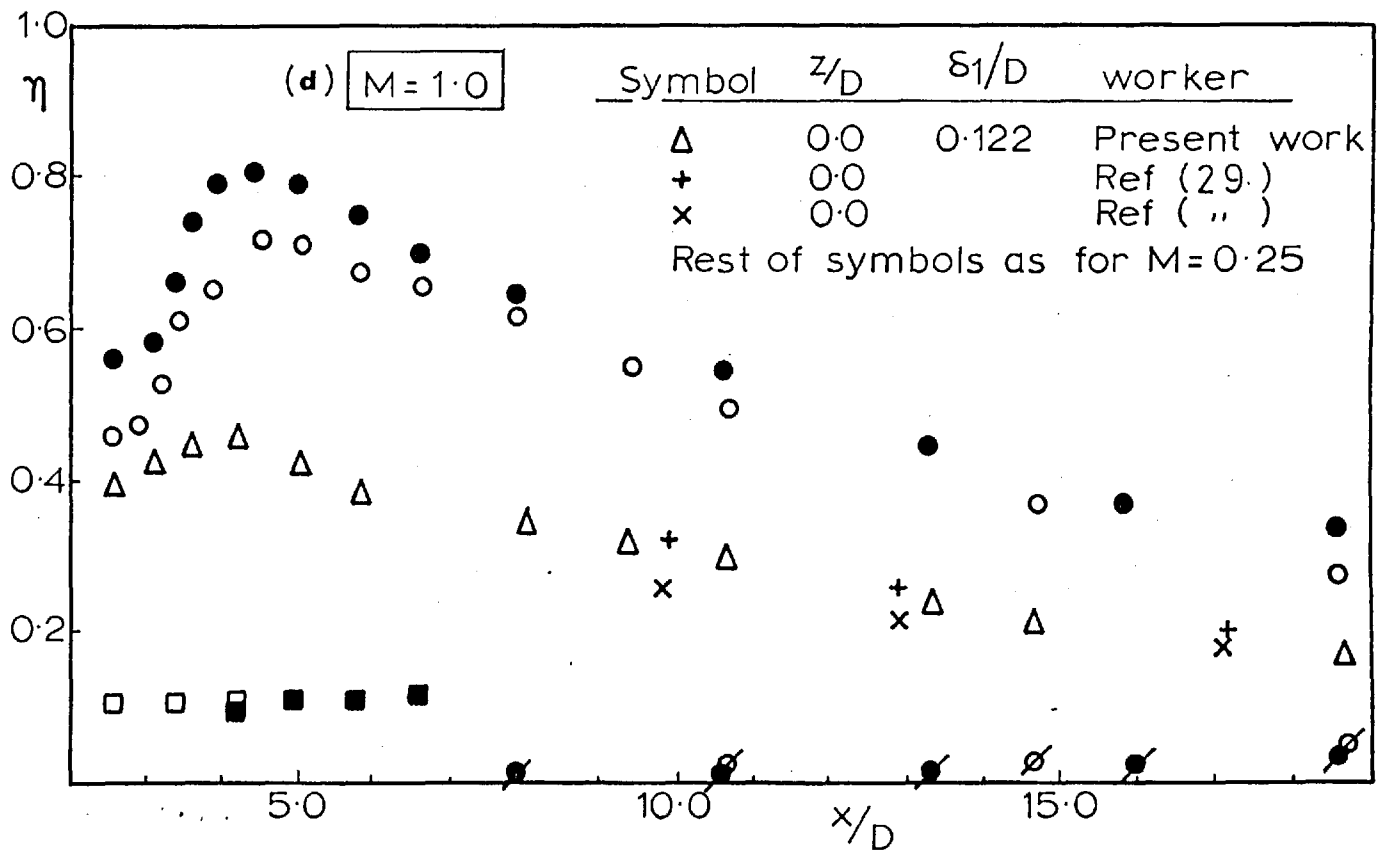
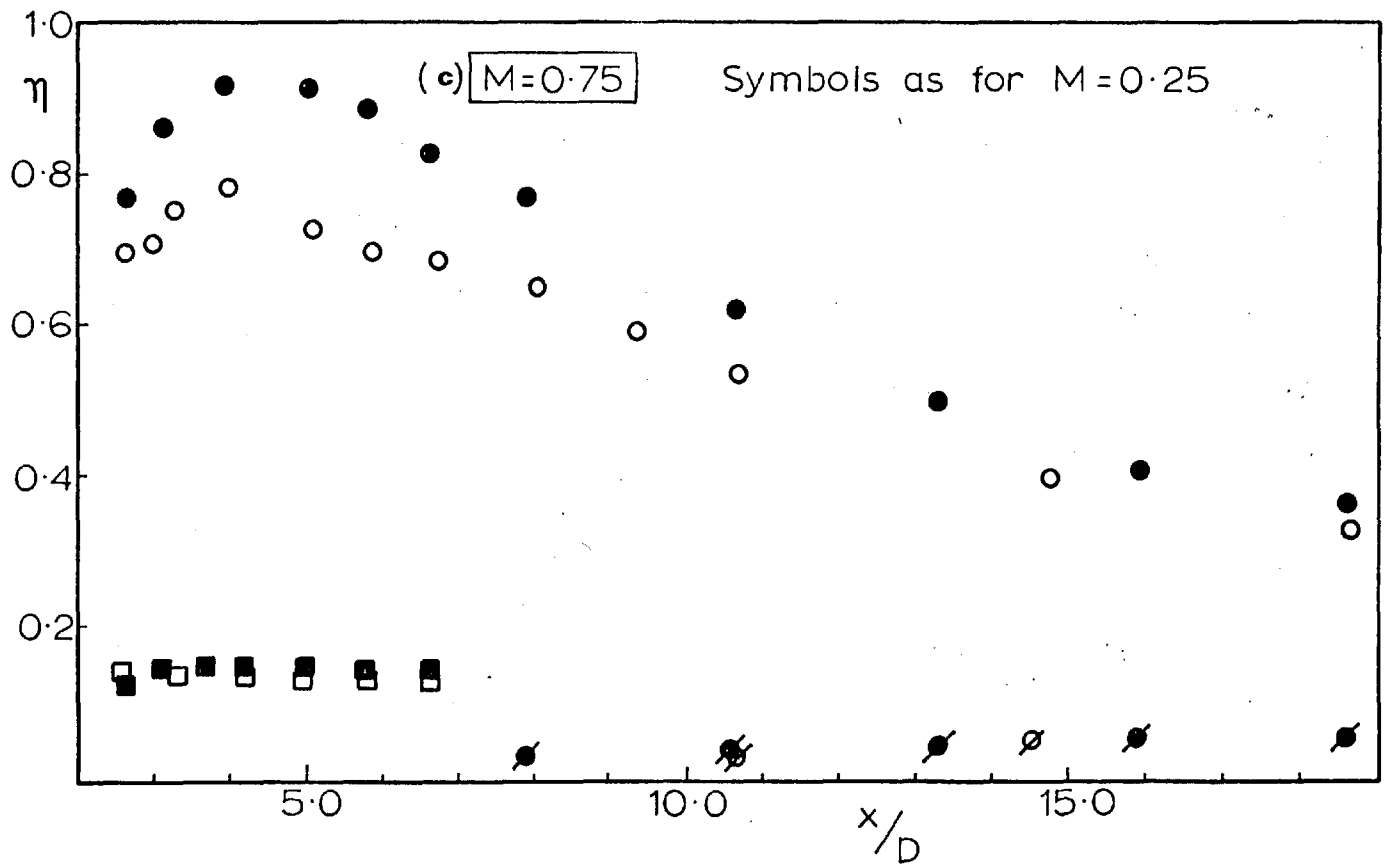


Fig. 44: Effect of boundary layer thickness on η , Row of holes

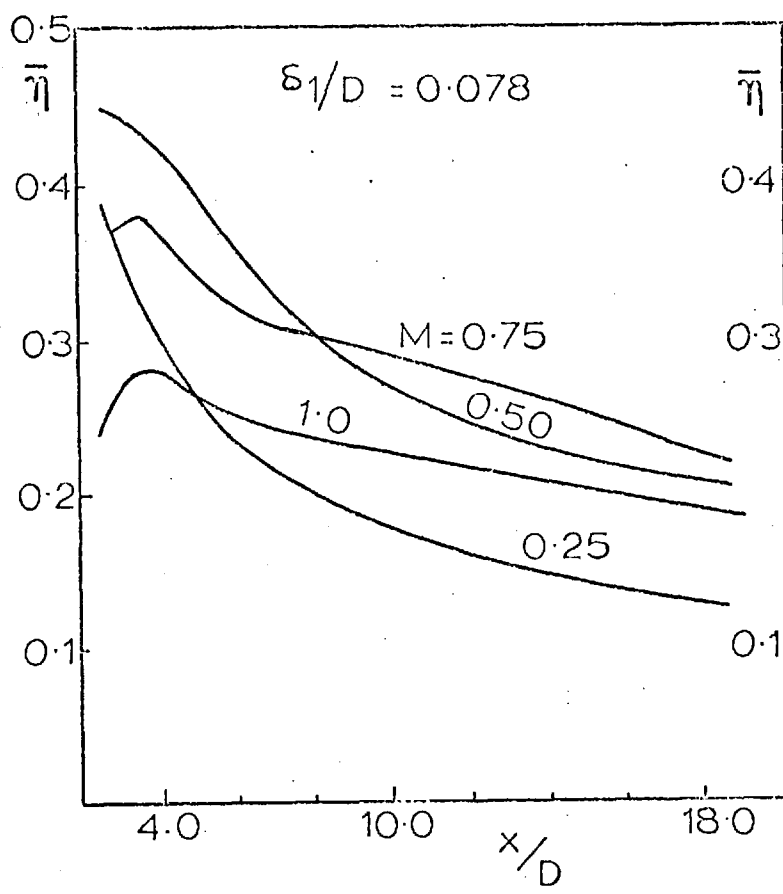


Fig. 45: Spanwise-averaged effectiveness

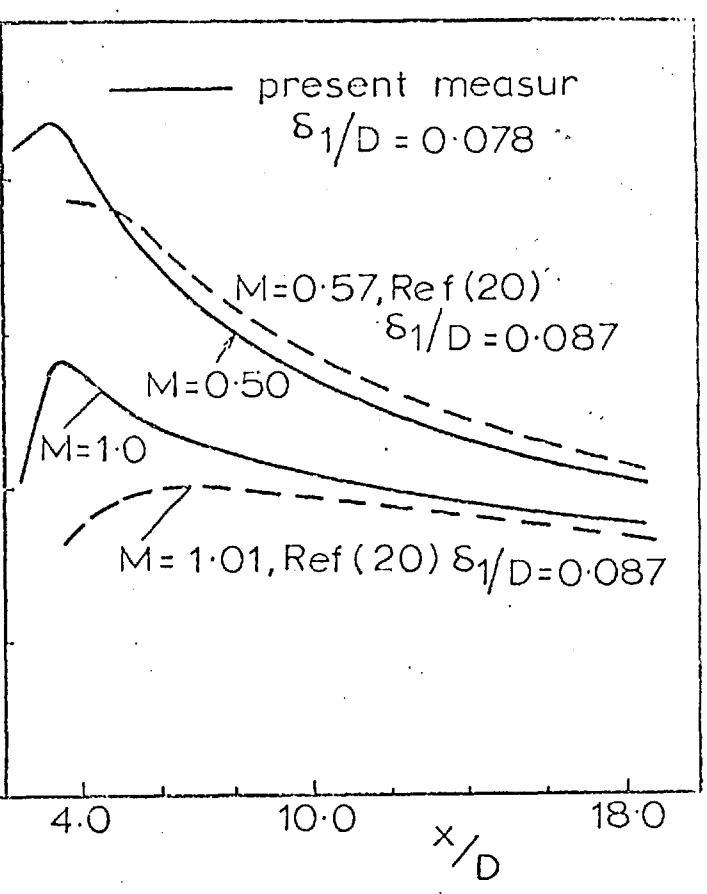


Fig. 46: Comparison of measurements with other sources



Fig. 47: Plan view of jets, double row, $M = 0.50$

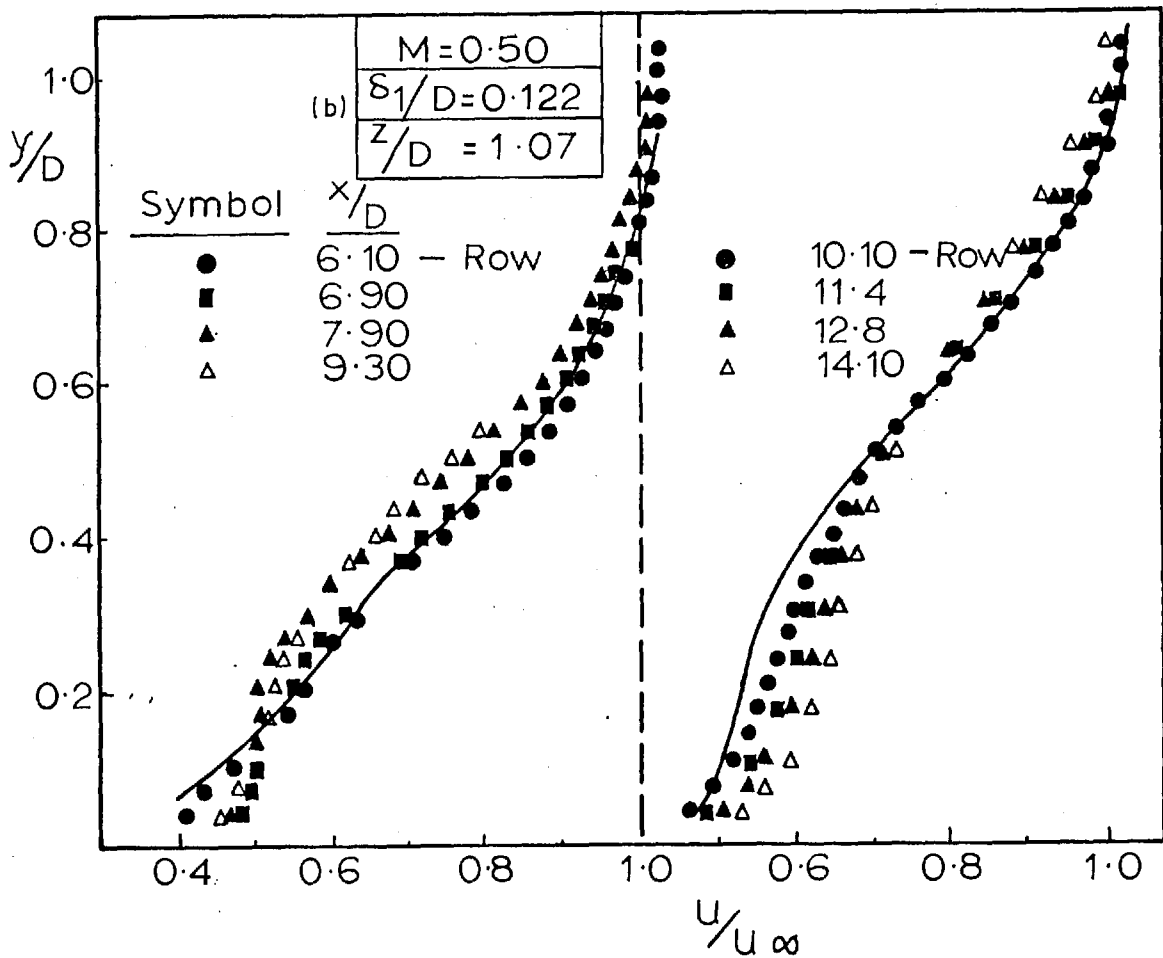
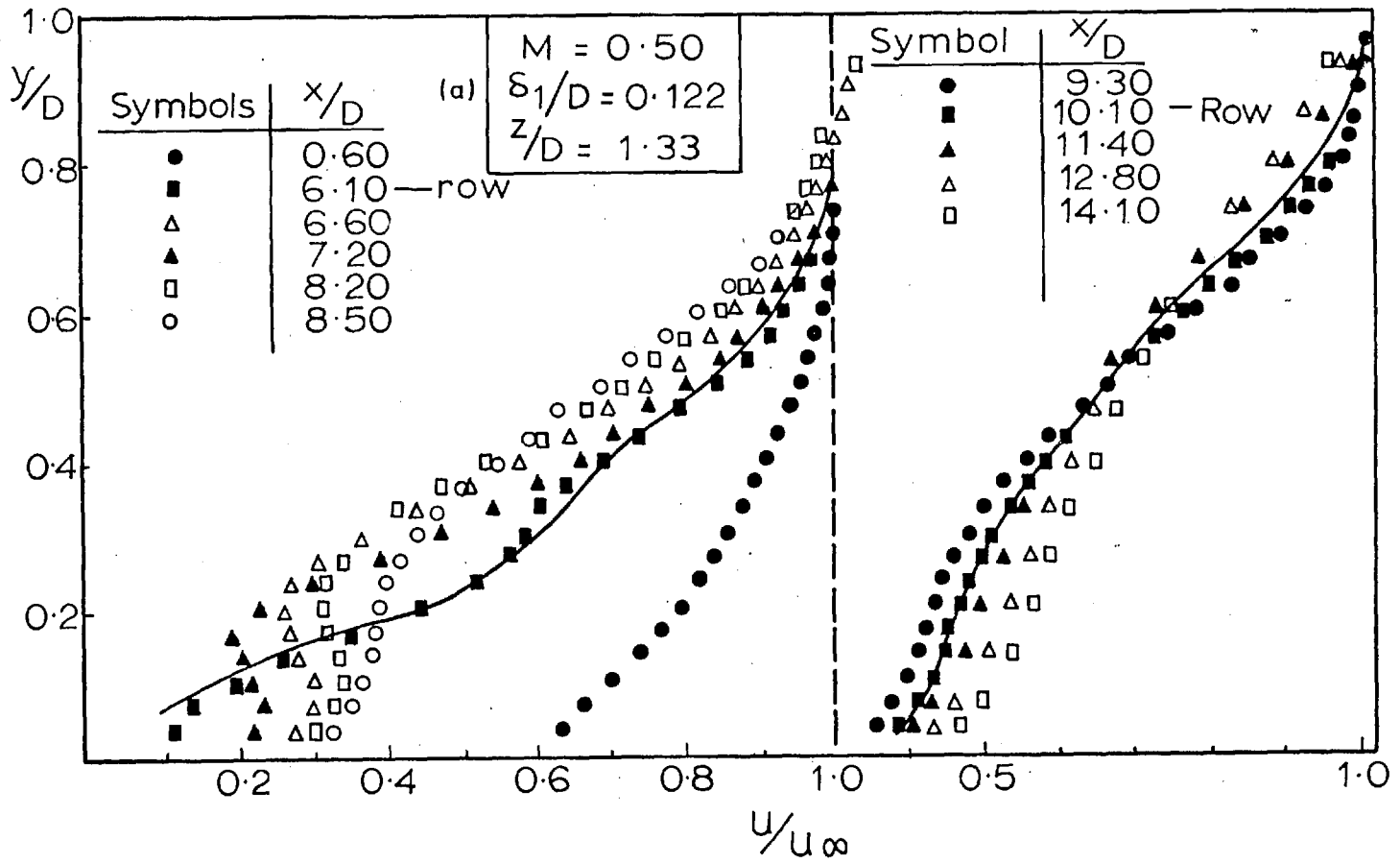


Fig. 48: Streamwise velocity profiles, double row, $M=0.50$

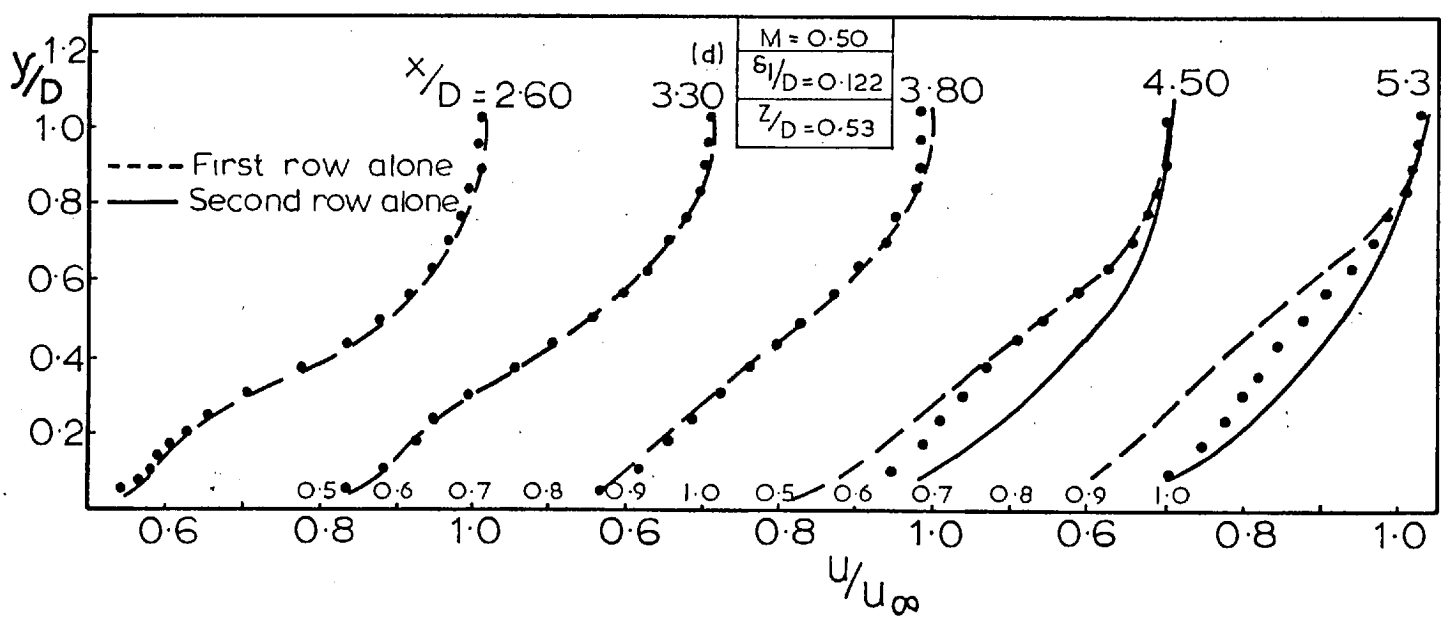
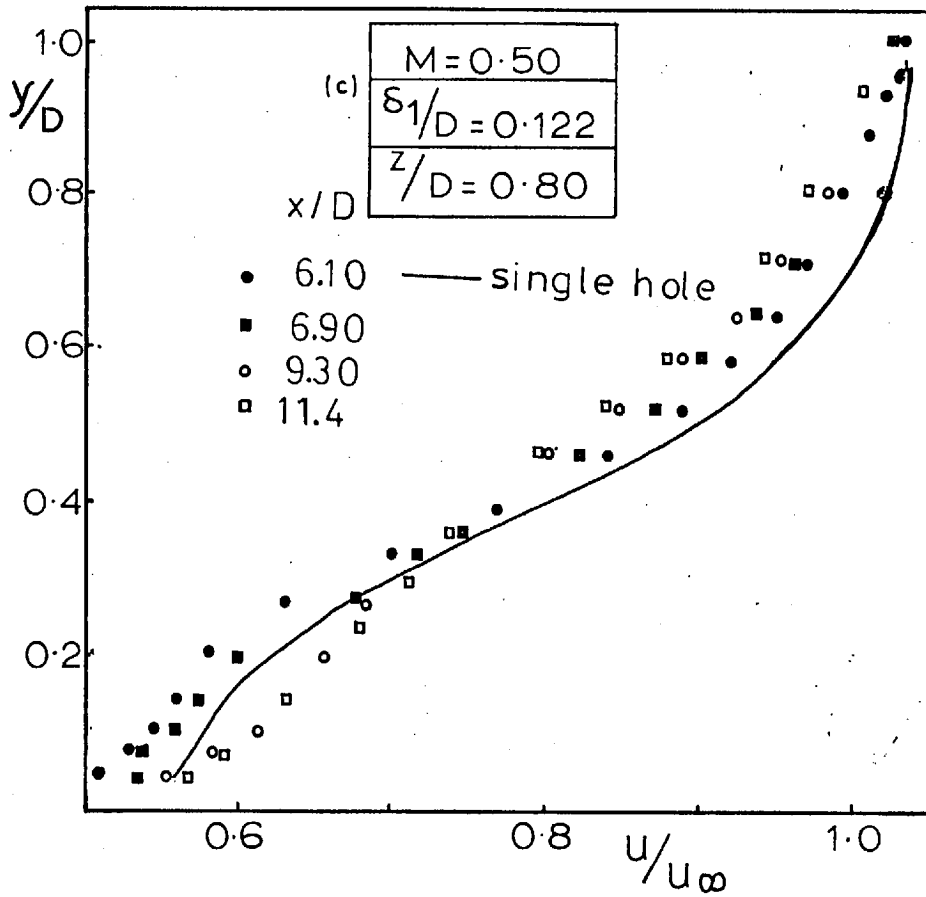


Fig. 48: Streamwise velocity profiles, double row, $M=0.50$

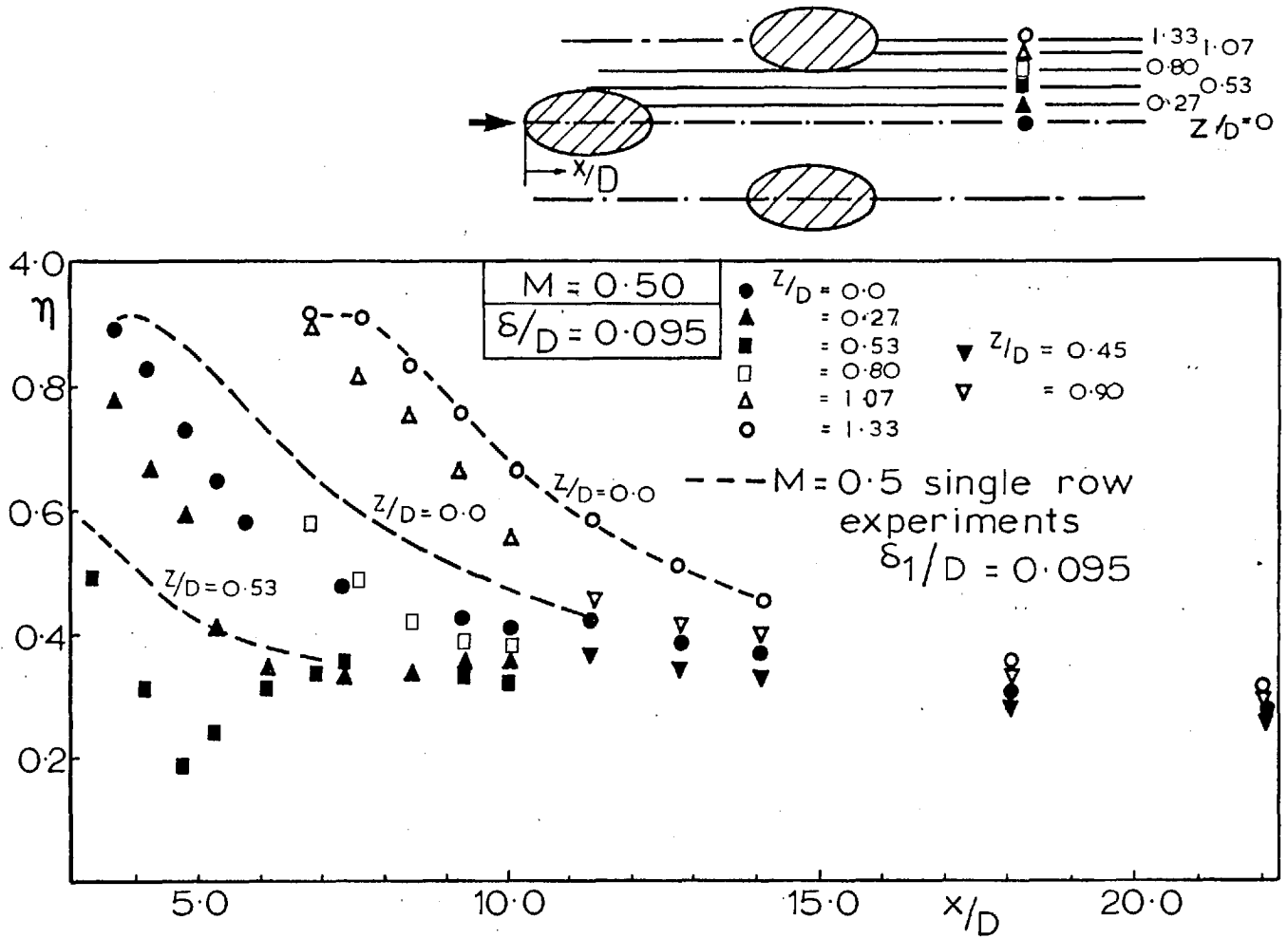


Fig. 49: Film cooling effectiveness, double row, $M=0.50$

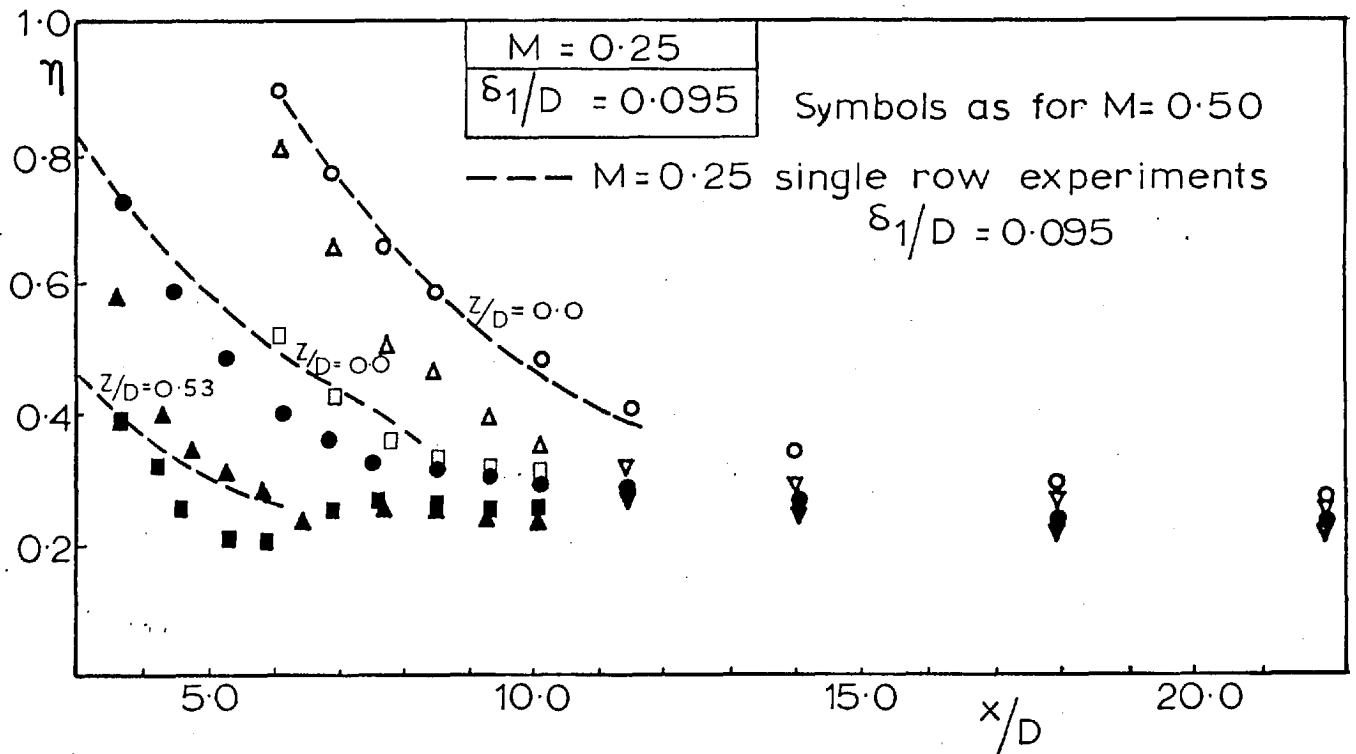


Fig. 50: Film cooling effectiveness, double row, $M=0.25$

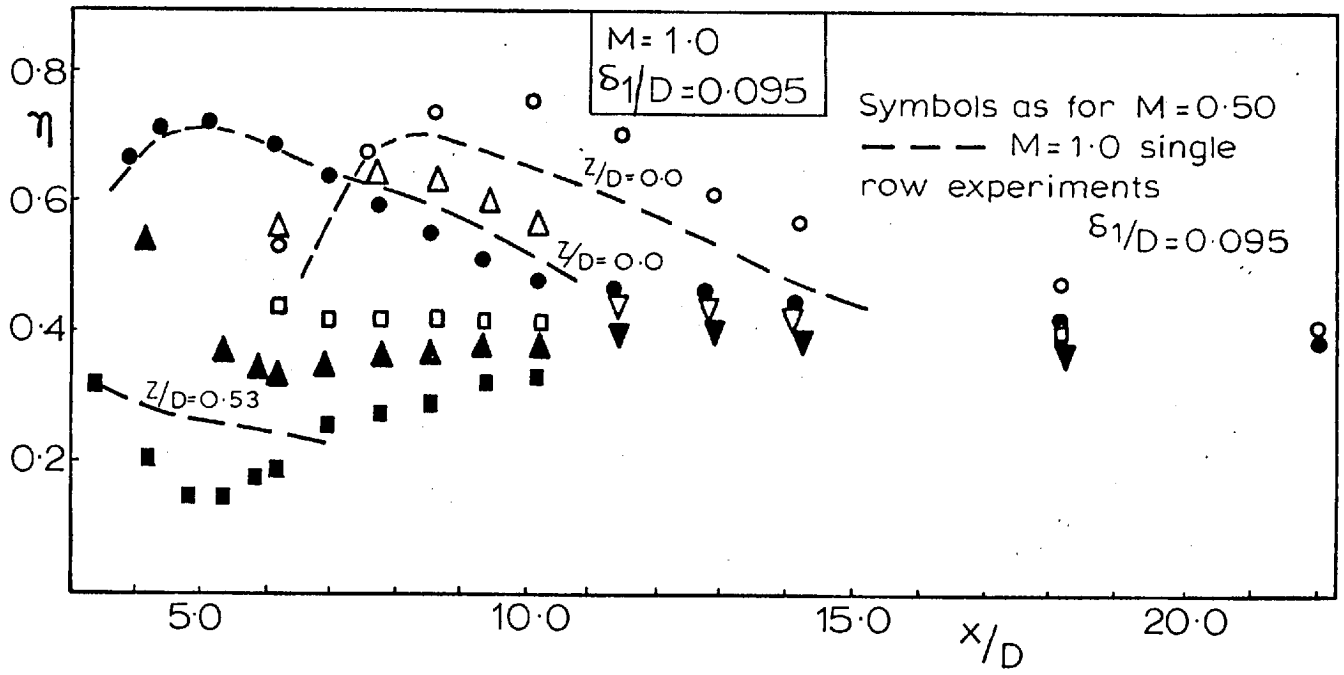


Fig. 51: Film cooling effectiveness, double row, $M=1.0$

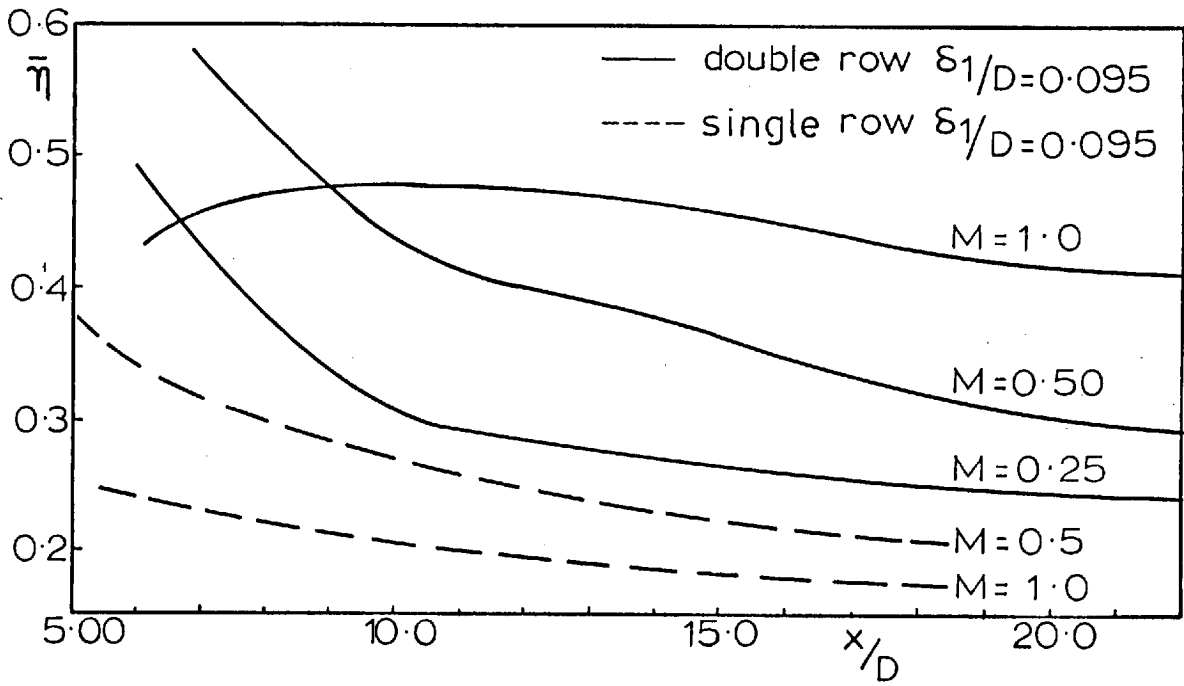


Fig. 52: Laterally averaged wall effectiveness, double row

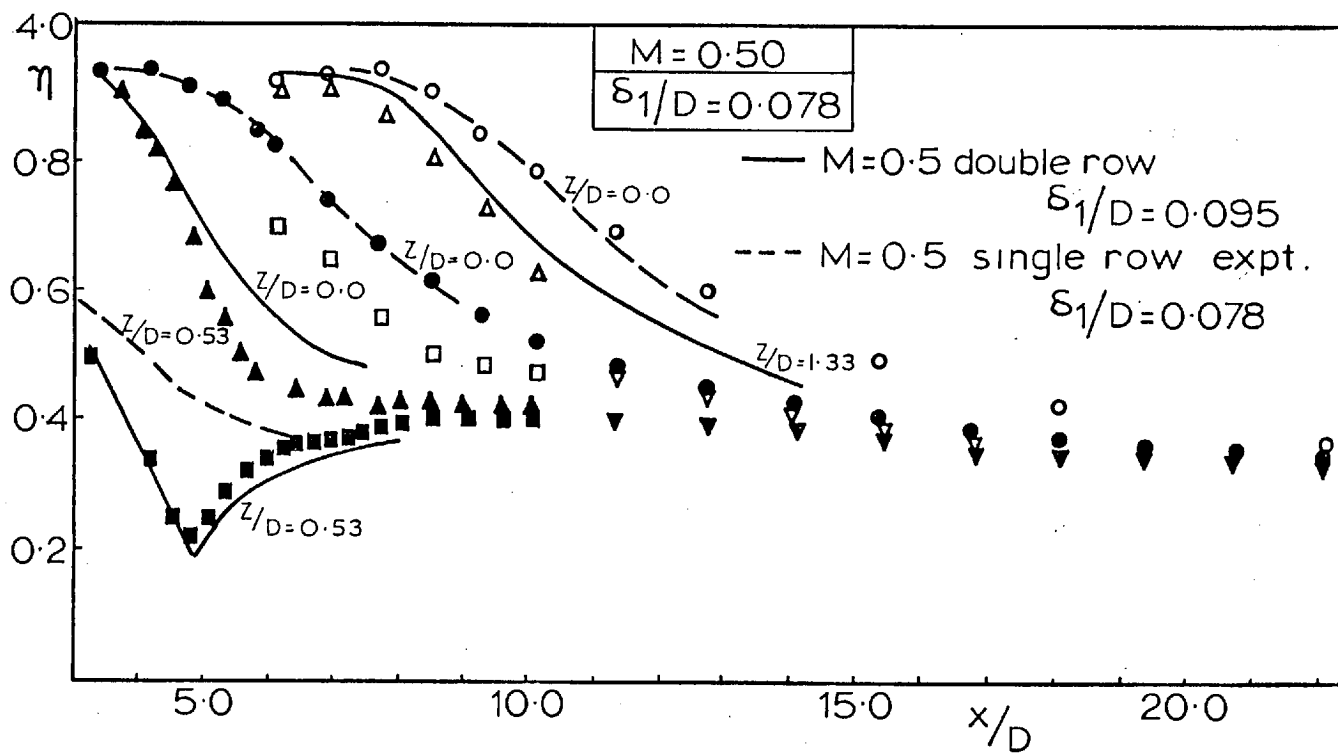


Fig. 53: Film cooling effectiveness, double row, $M=0.50$

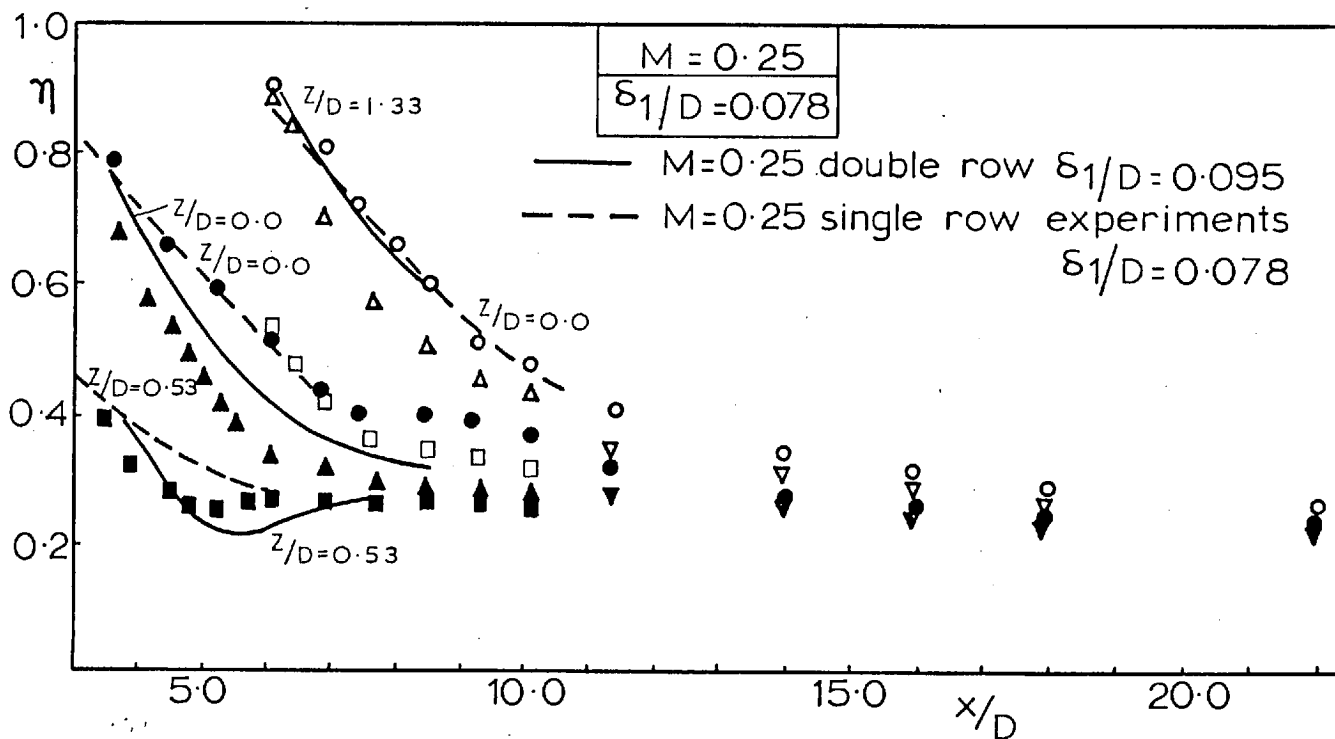


Fig. 54: Film cooling effectiveness, double row, $M=0.25$

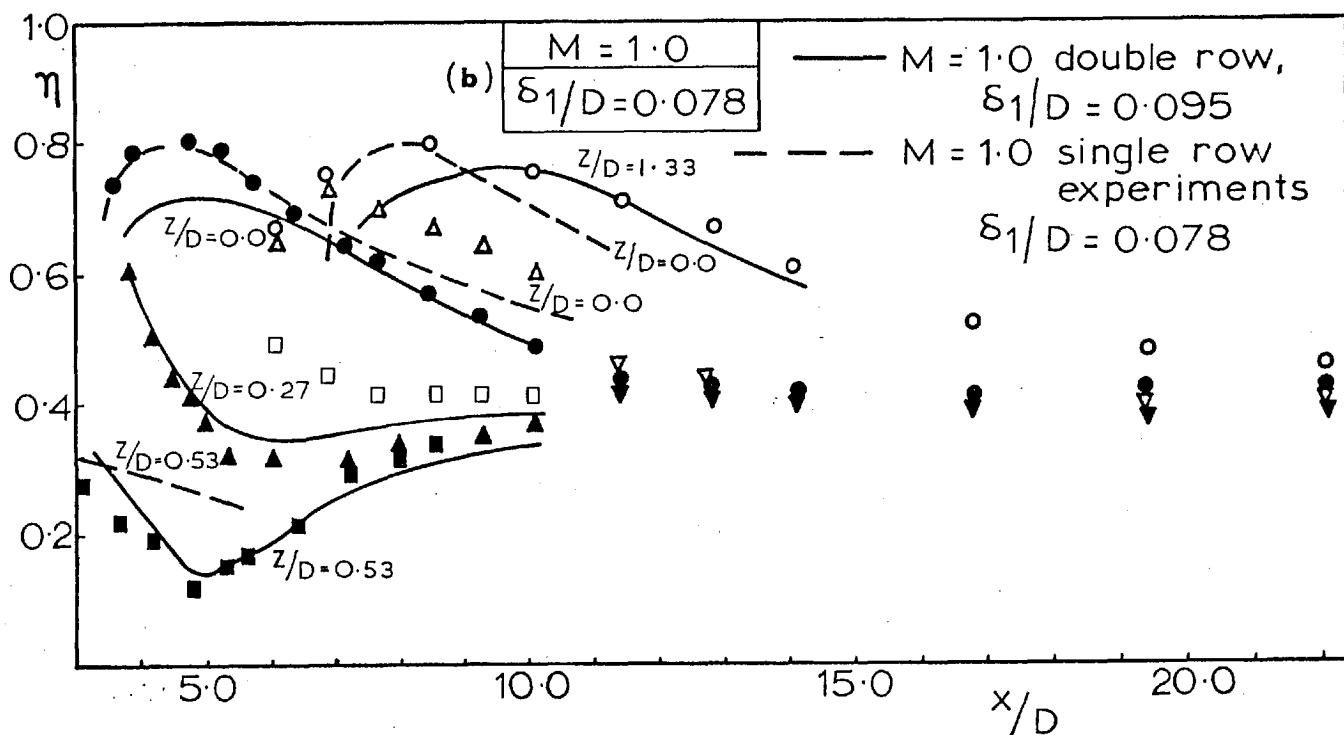
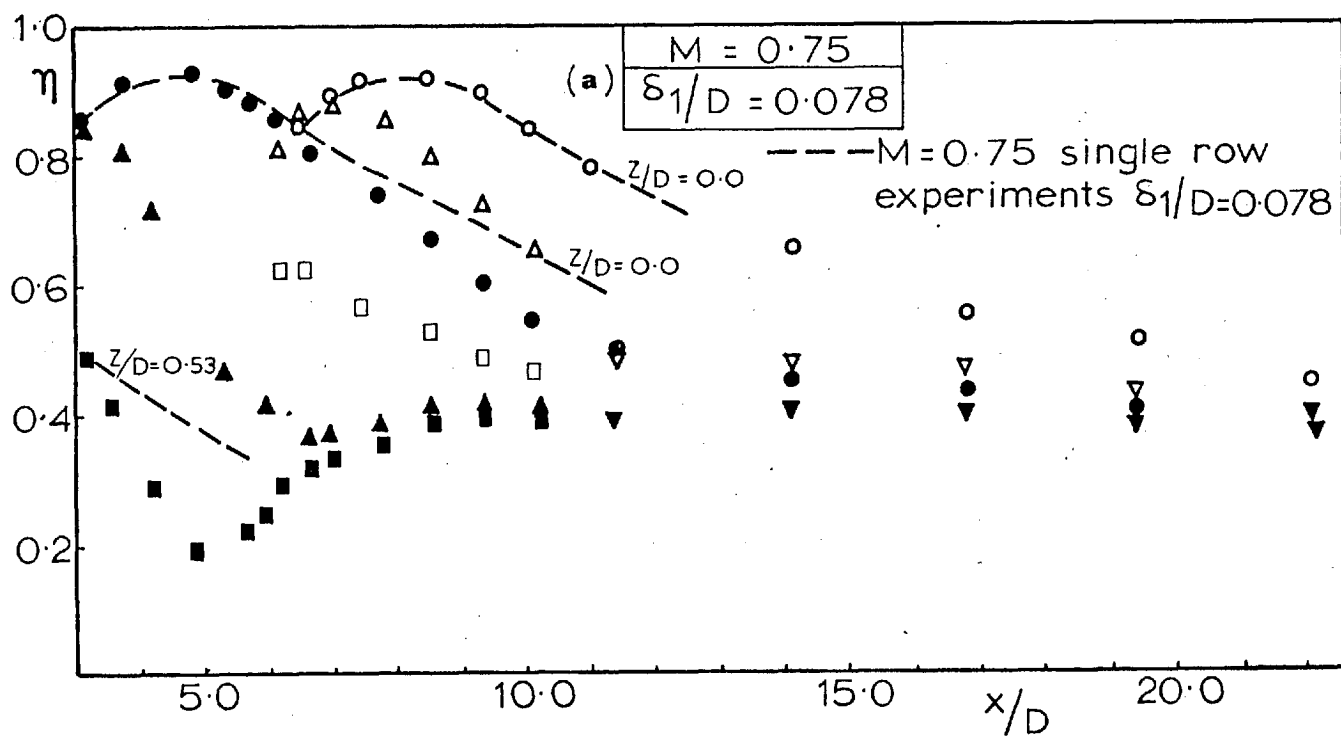


Fig. 55: Film cooling effectiveness, double row

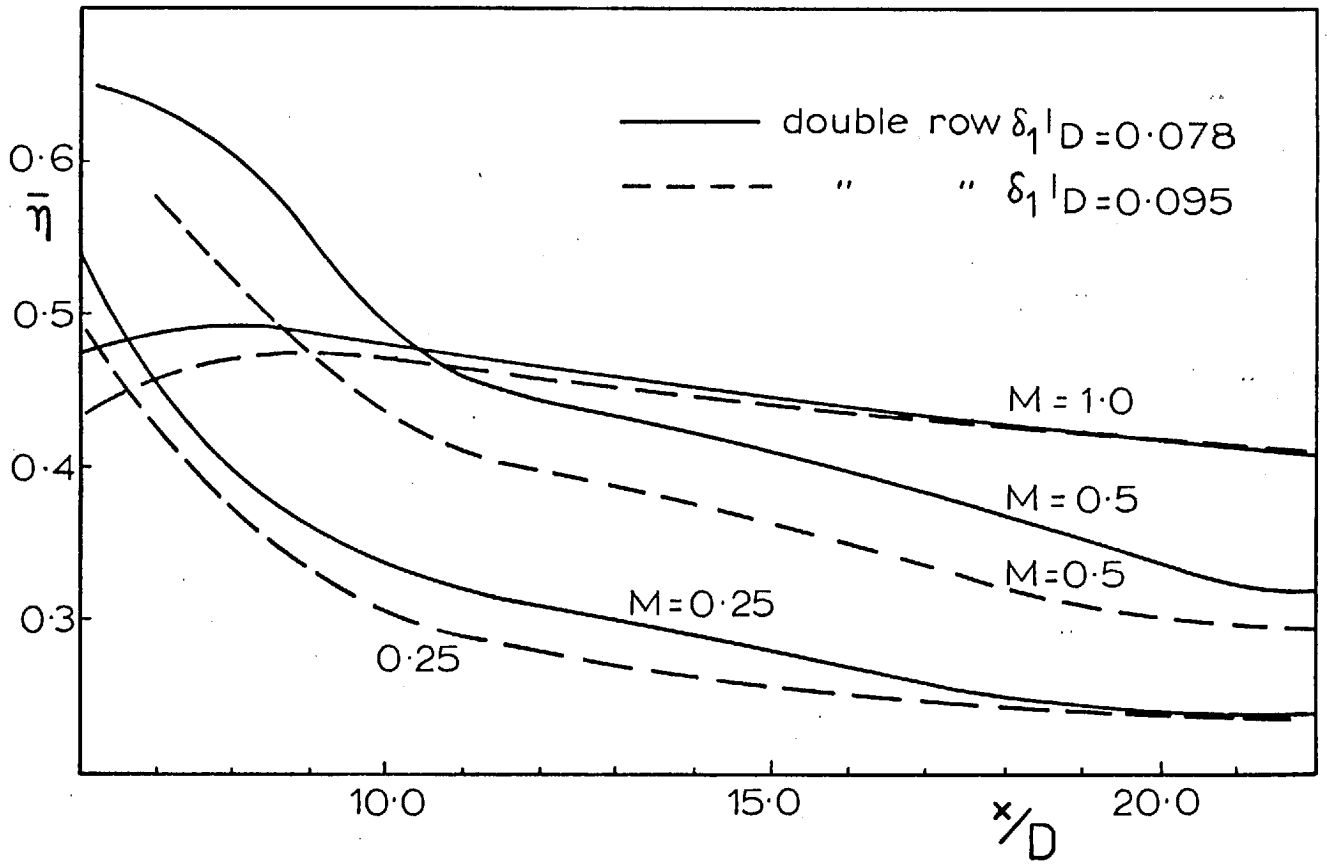
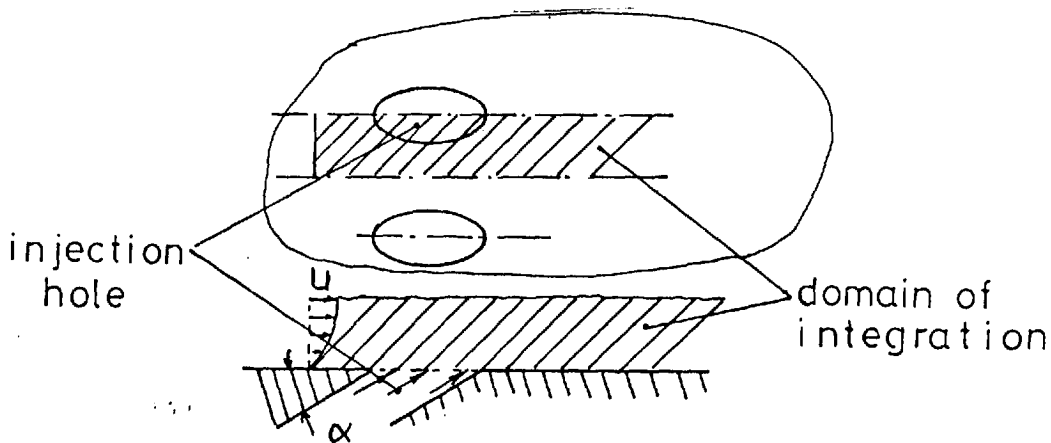
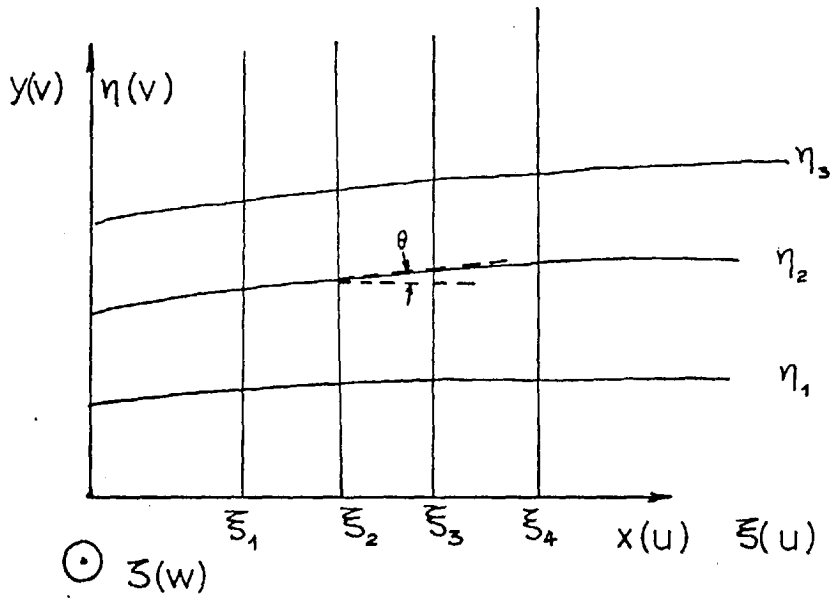
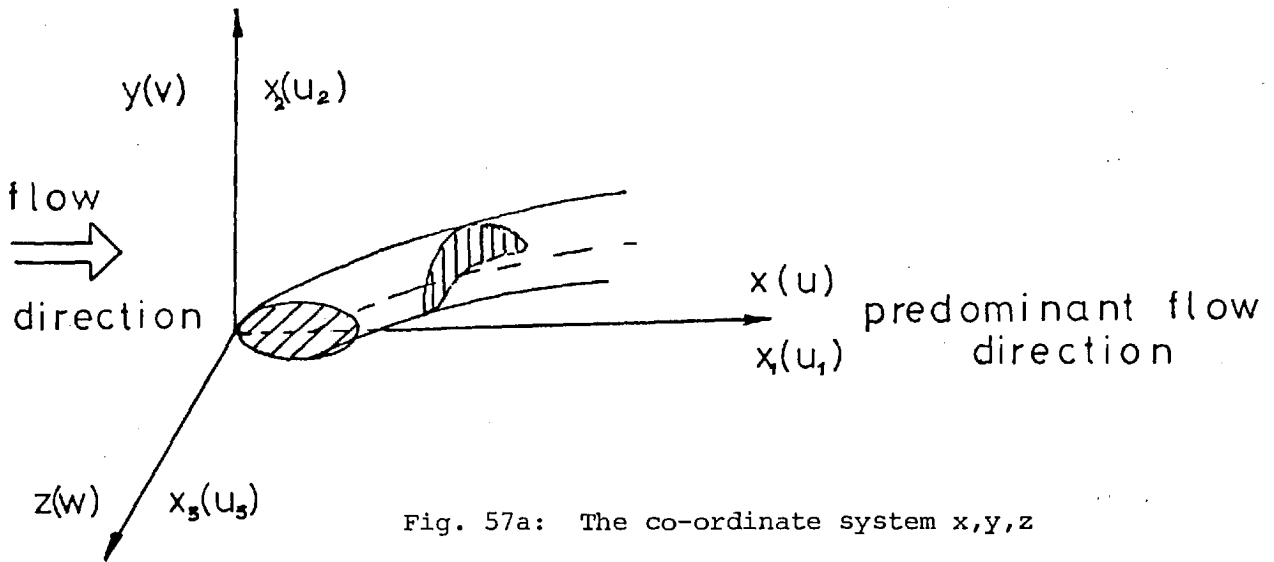


Fig. 56: Laterally averaged wall effectiveness, effect of boundary layer thickness, double row



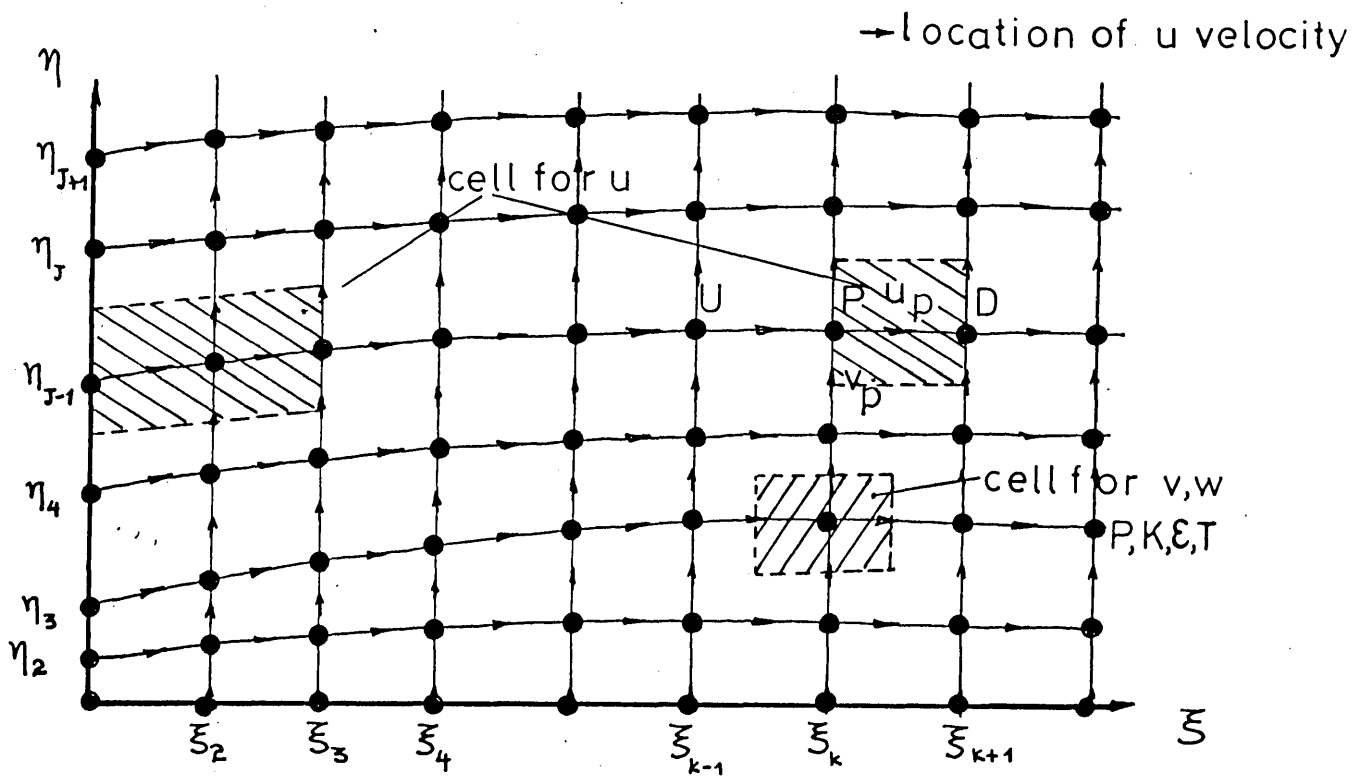
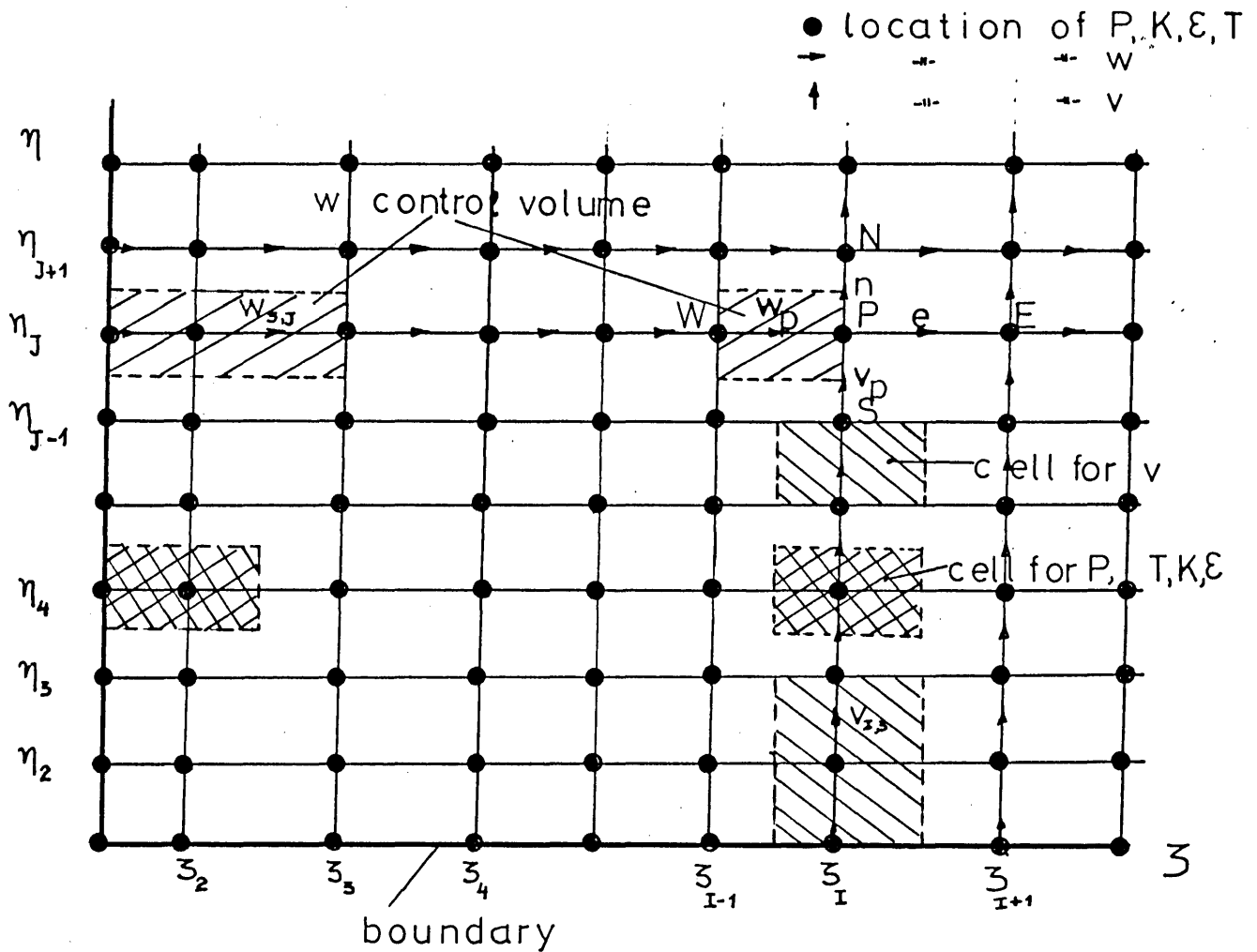


Fig. 57d: Location of dependent variables on the grid; control volumes

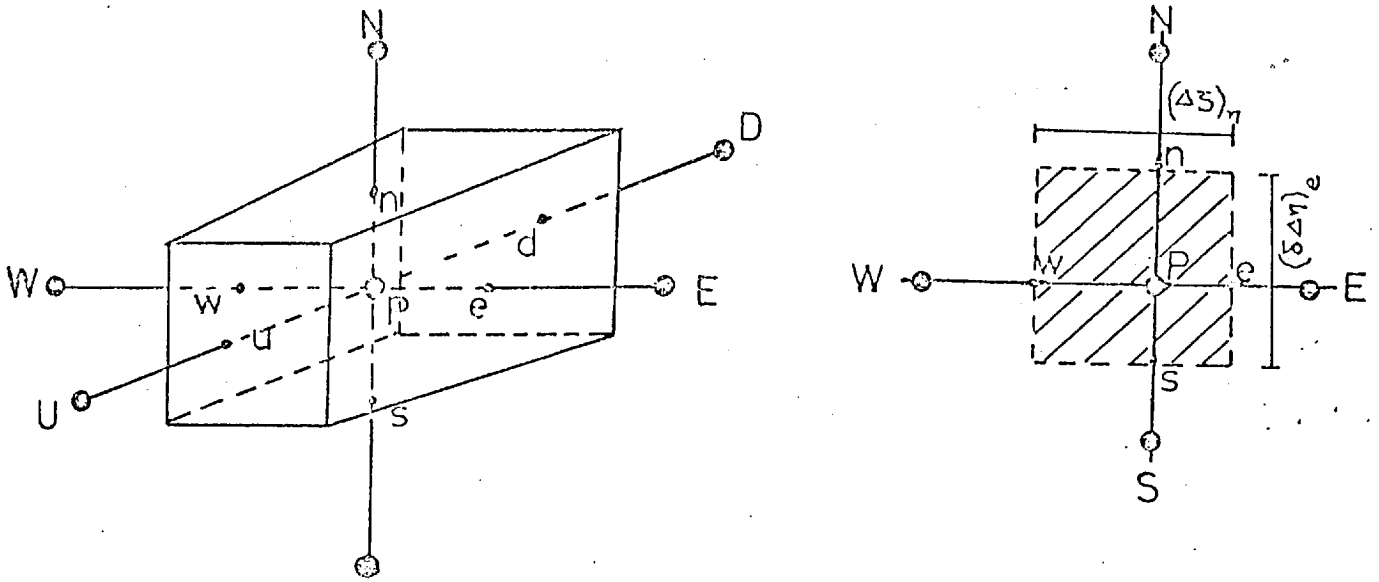


Fig. 57e: Control volume for continuity, T , k and ϵ

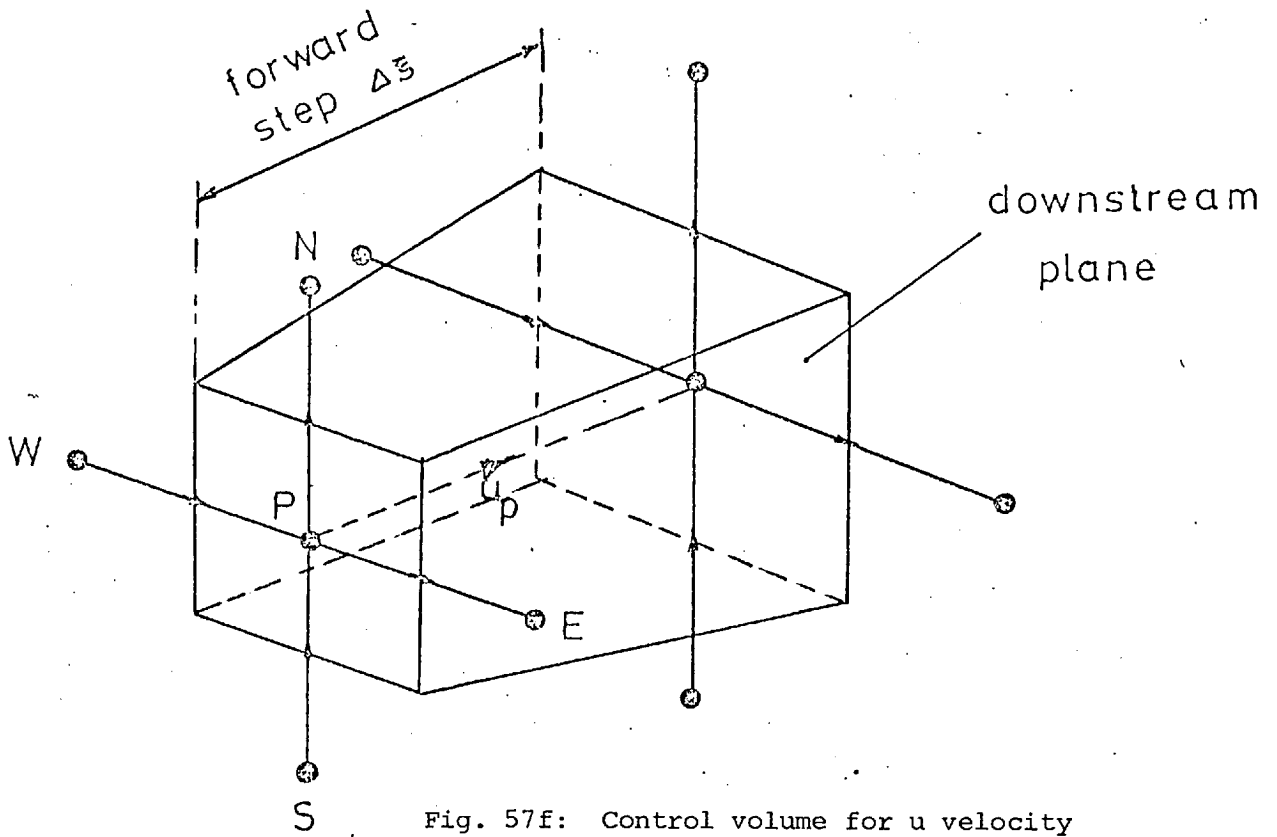


Fig. 57f: Control volume for u velocity

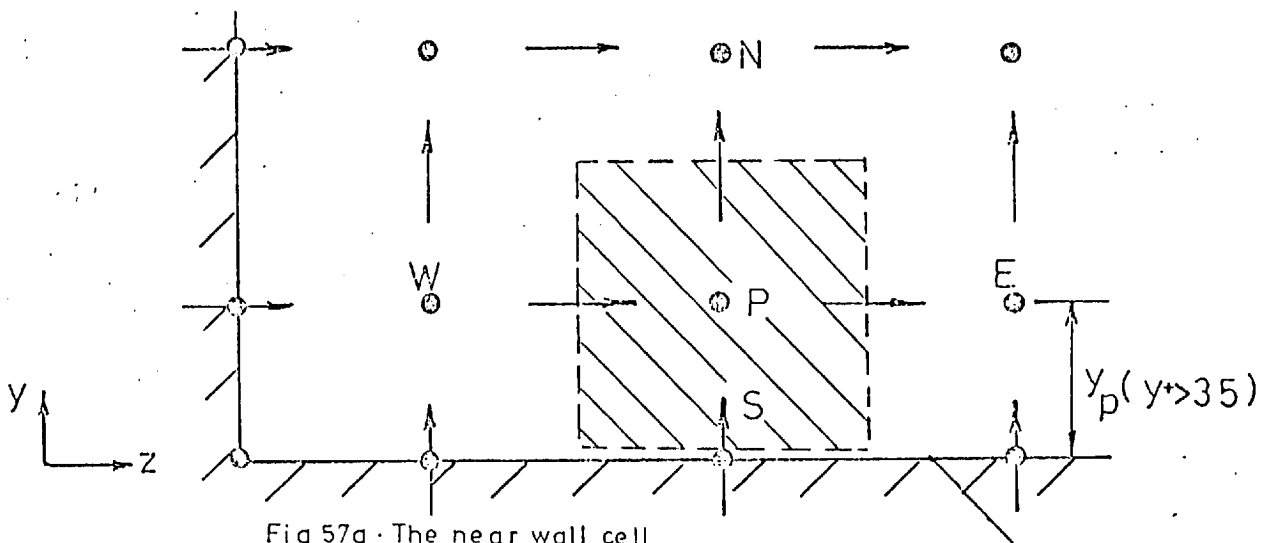


Fig. 57g: The near wall cell

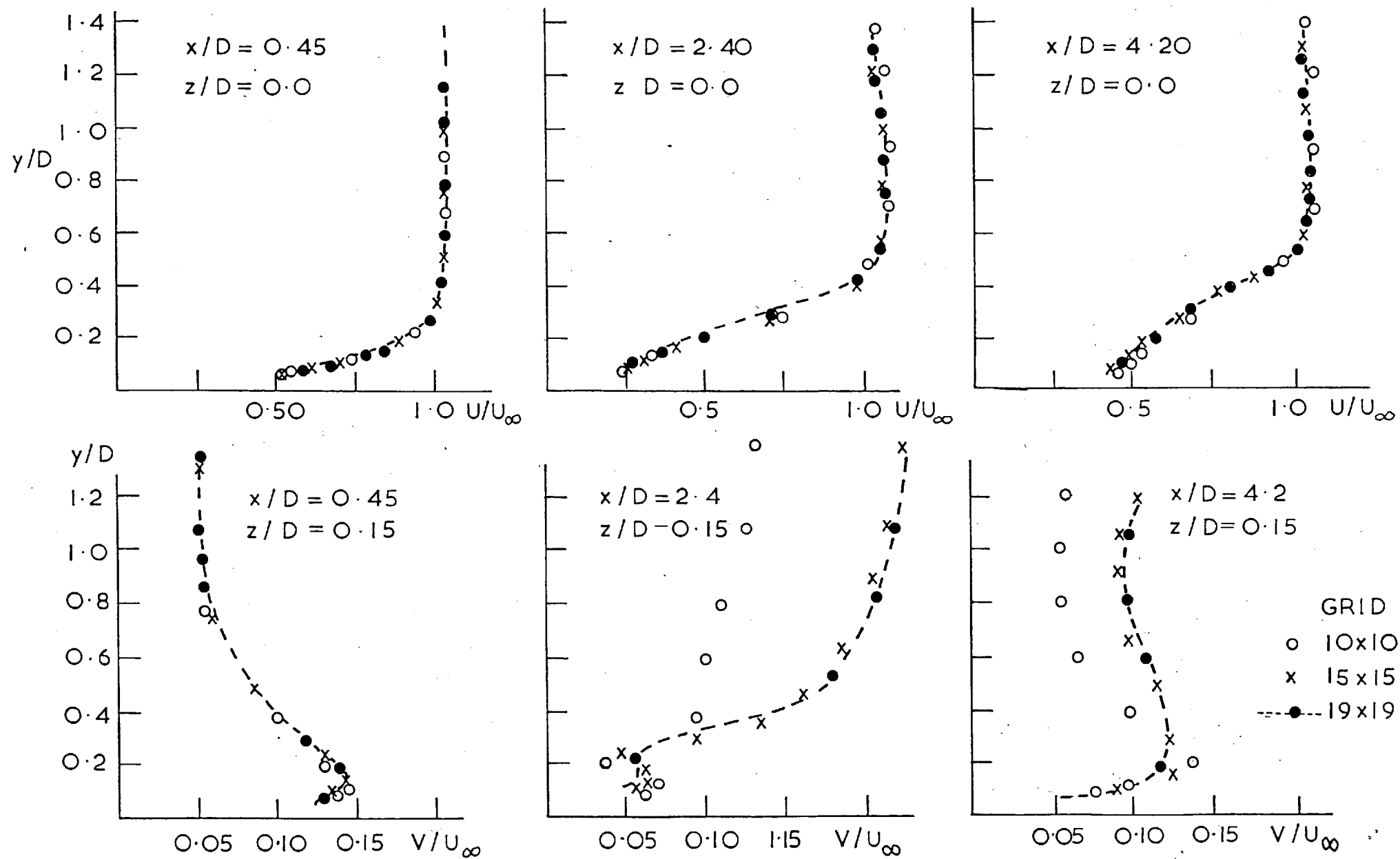


Fig. 58a : Grid dependence tests, $M=0.1$, $\alpha=90^\circ$, ref. (31)

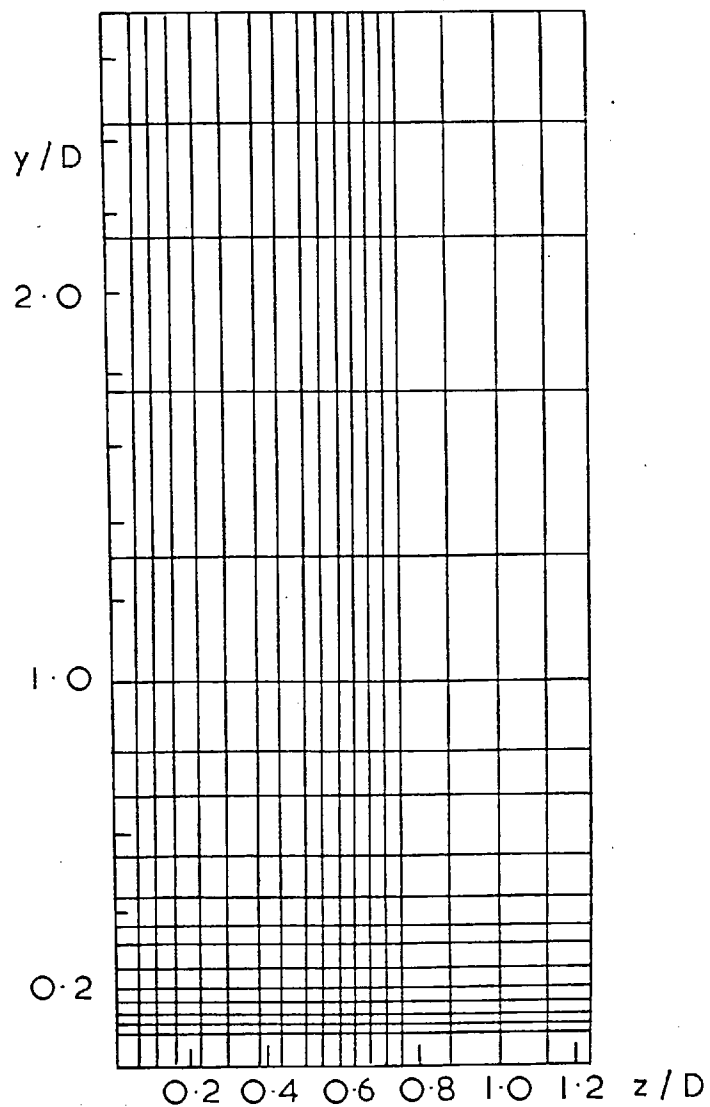


Fig. 58b : The grid distribution.

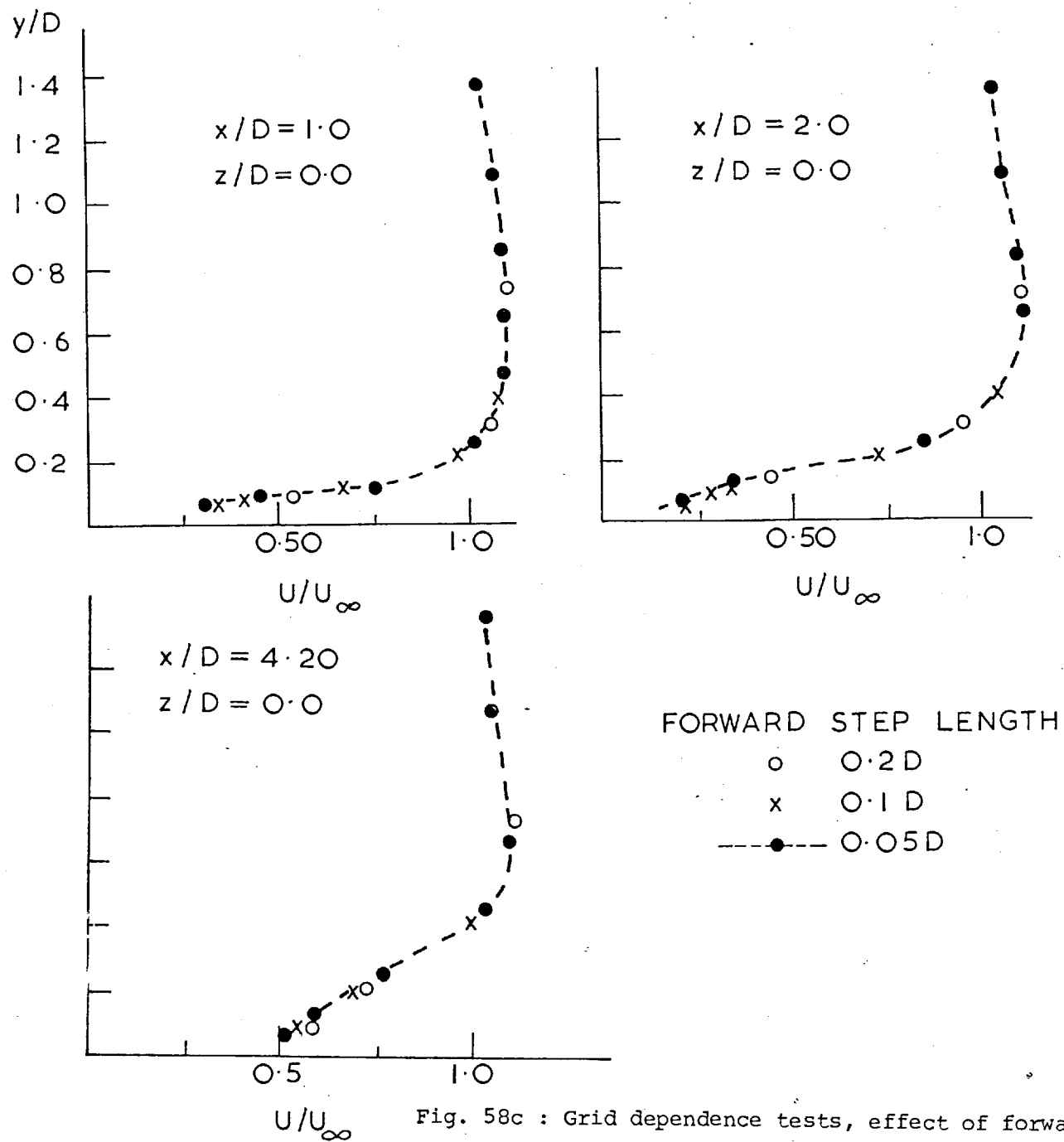


Fig. 58c : Grid dependence tests, effect of forward step size.

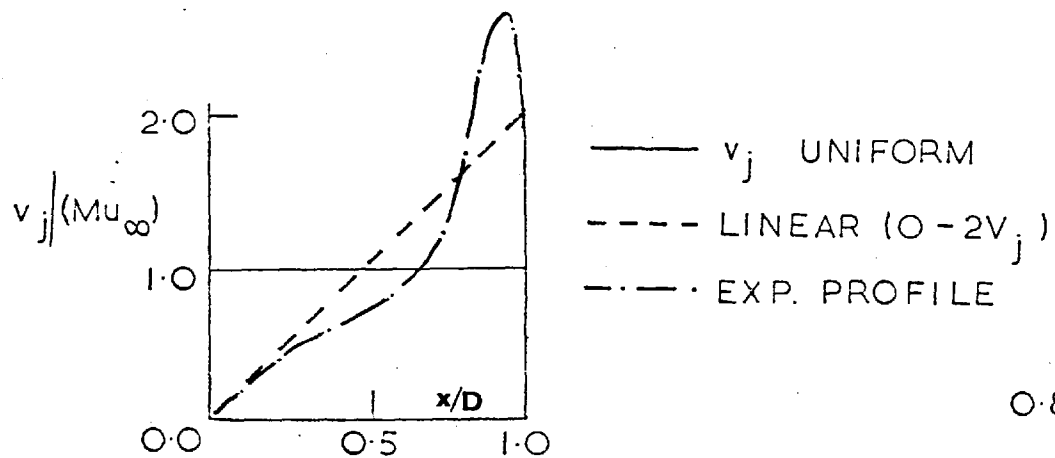


Fig. 58d : Secondary injection velocity profiles for numerical exploration.

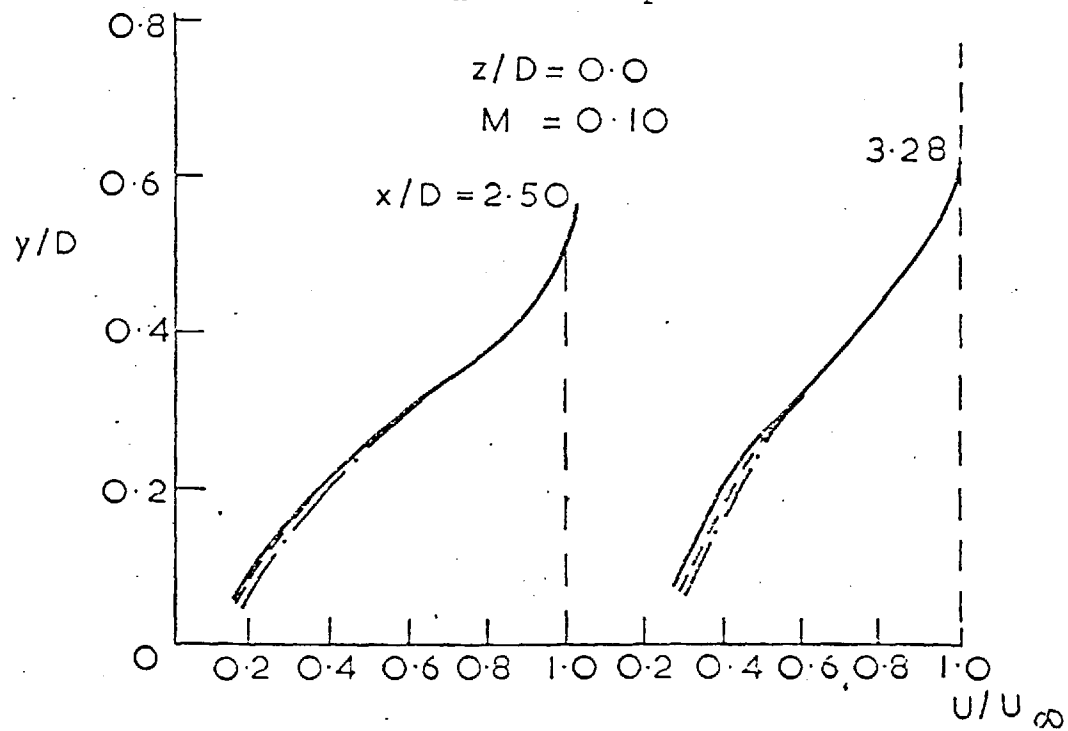


Fig. 58e : Effect of injection velocity profile on

$u, IC, \alpha=90^\circ, M=0.1.$

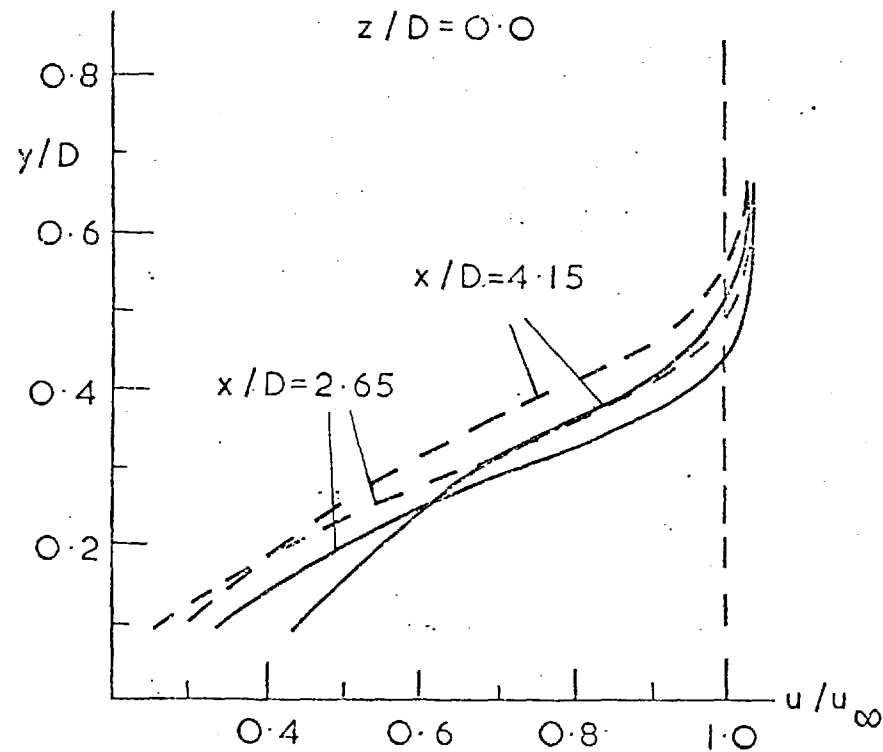


Fig. 59a : Effect of non-isotropic ($k-\epsilon$) on u .

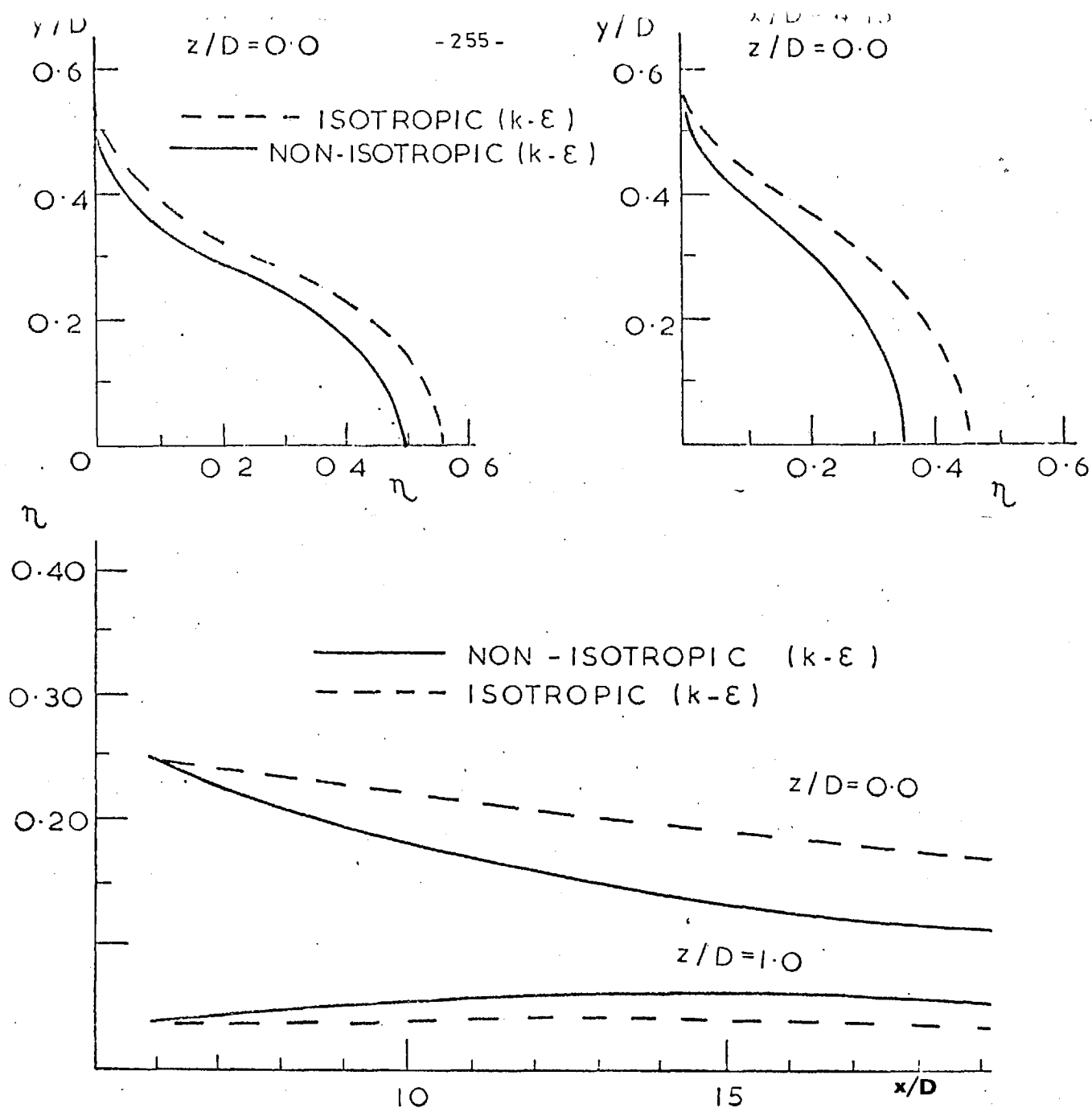


Fig. 59b : Effect of non-isotropic $(k-\epsilon)$ on η .

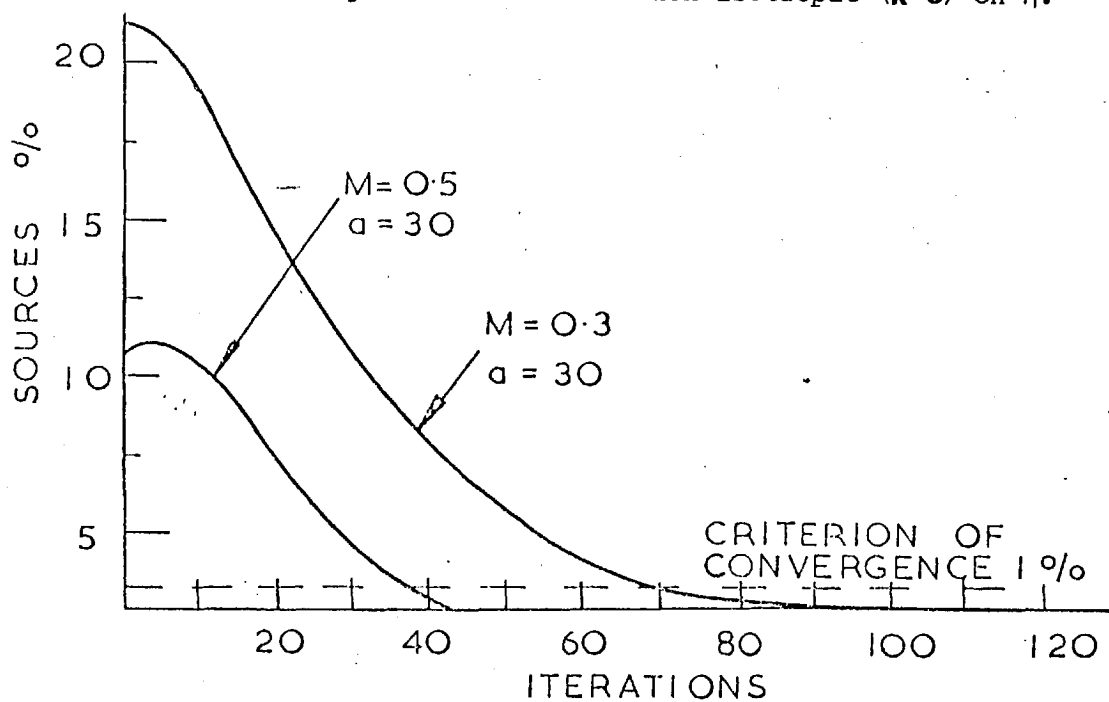
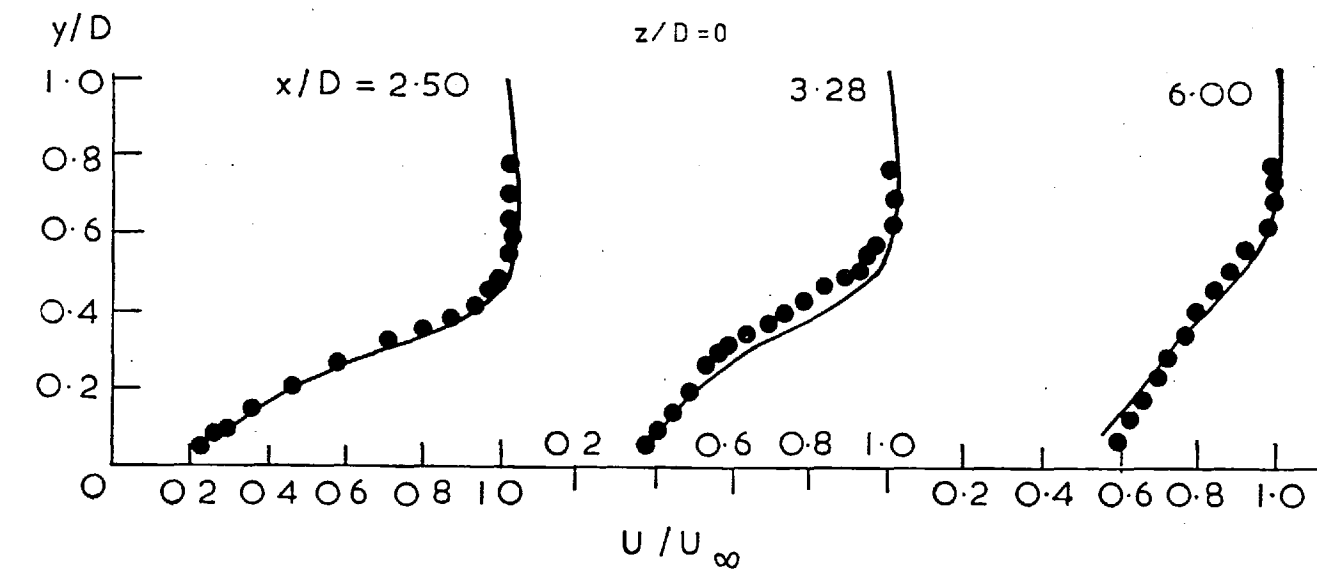
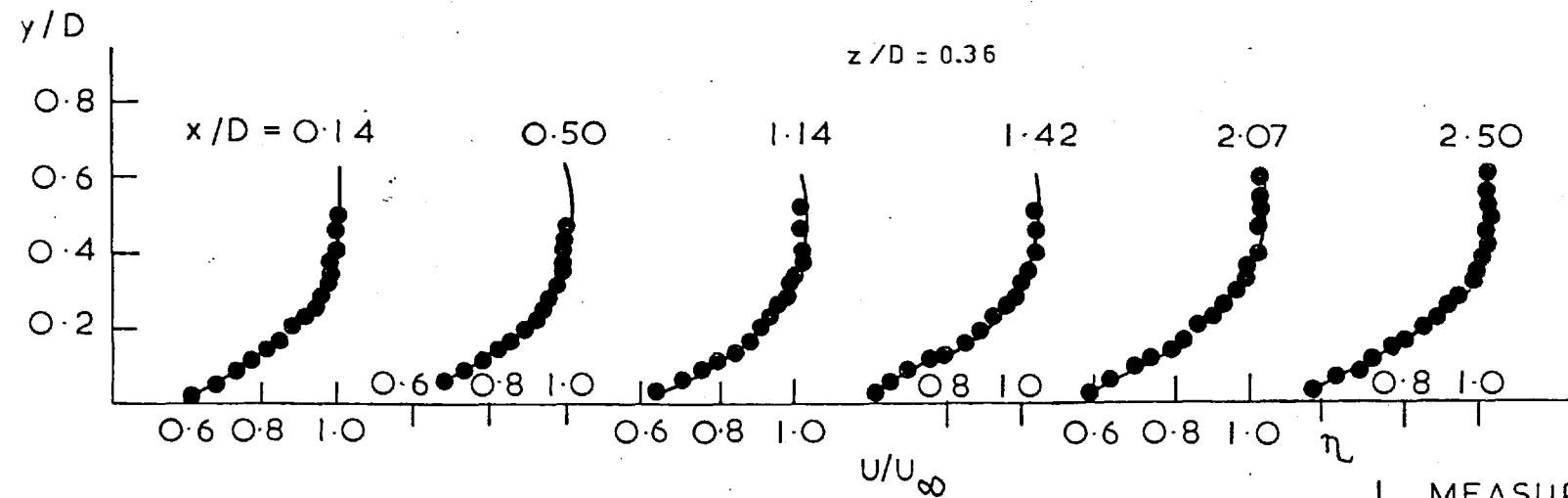


Fig. 60 : Rate of convergence of the semi-elliptic procedure.



• MEASUREMENTS, I.C. — PREDICTIONS

Fig. 6la : Predicted velocity profiles, comparison
with measurements I.C, $\alpha=90^\circ$, $M=0.046$

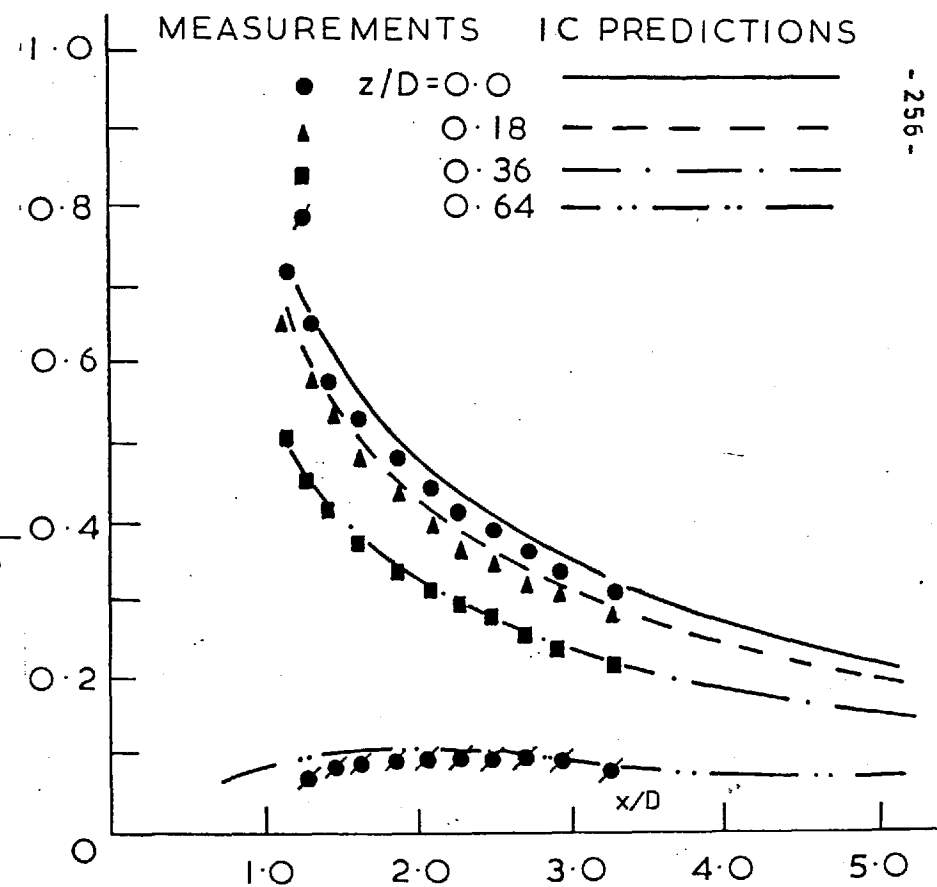


Fig. 6lb : Predicted film cooling effectiveness

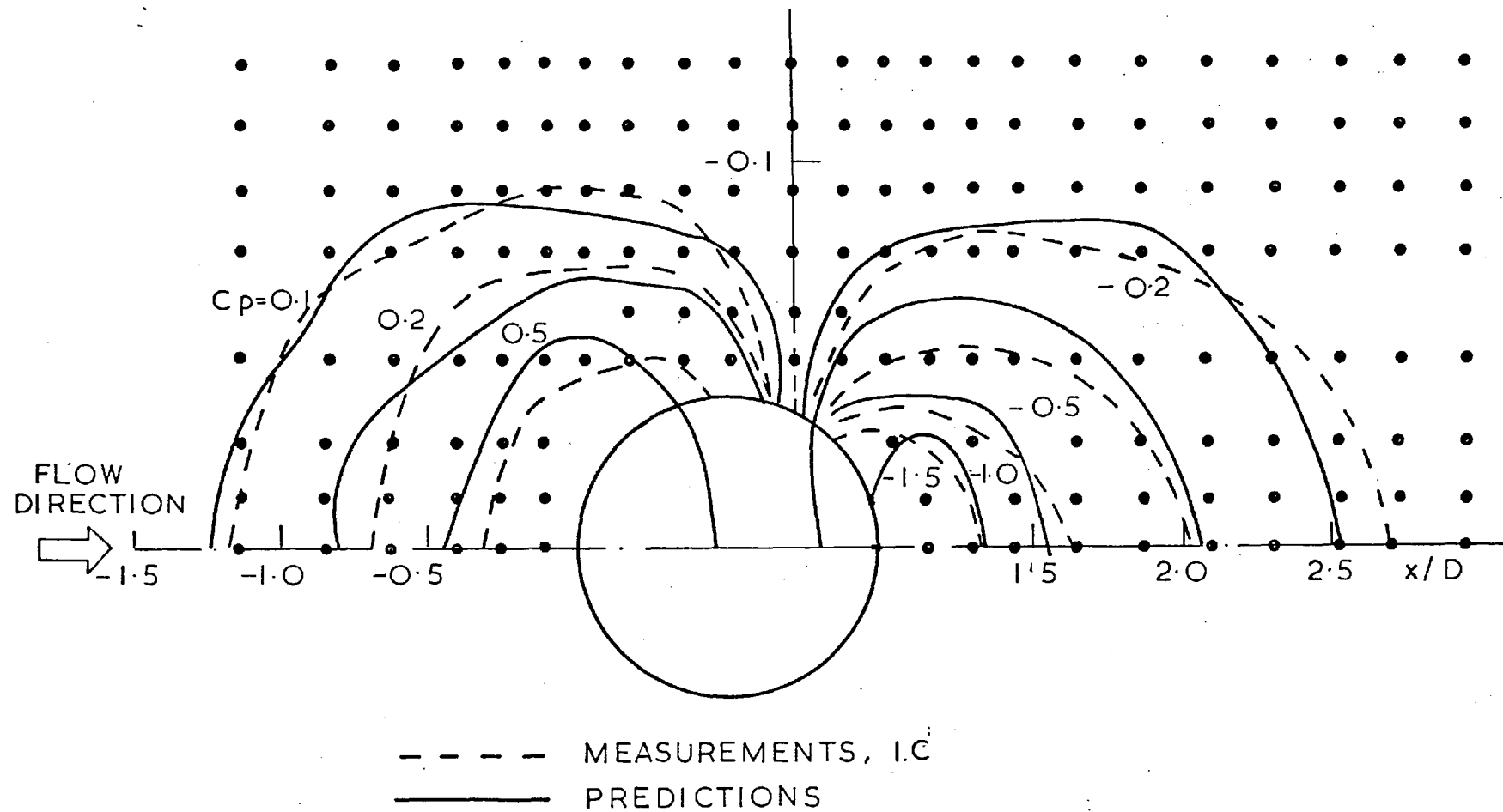


Fig. 62a : Predicted surface pressure distribution
 comparison with measurements, IC, $\alpha=90^\circ$,
 $M=0.1$, IC.

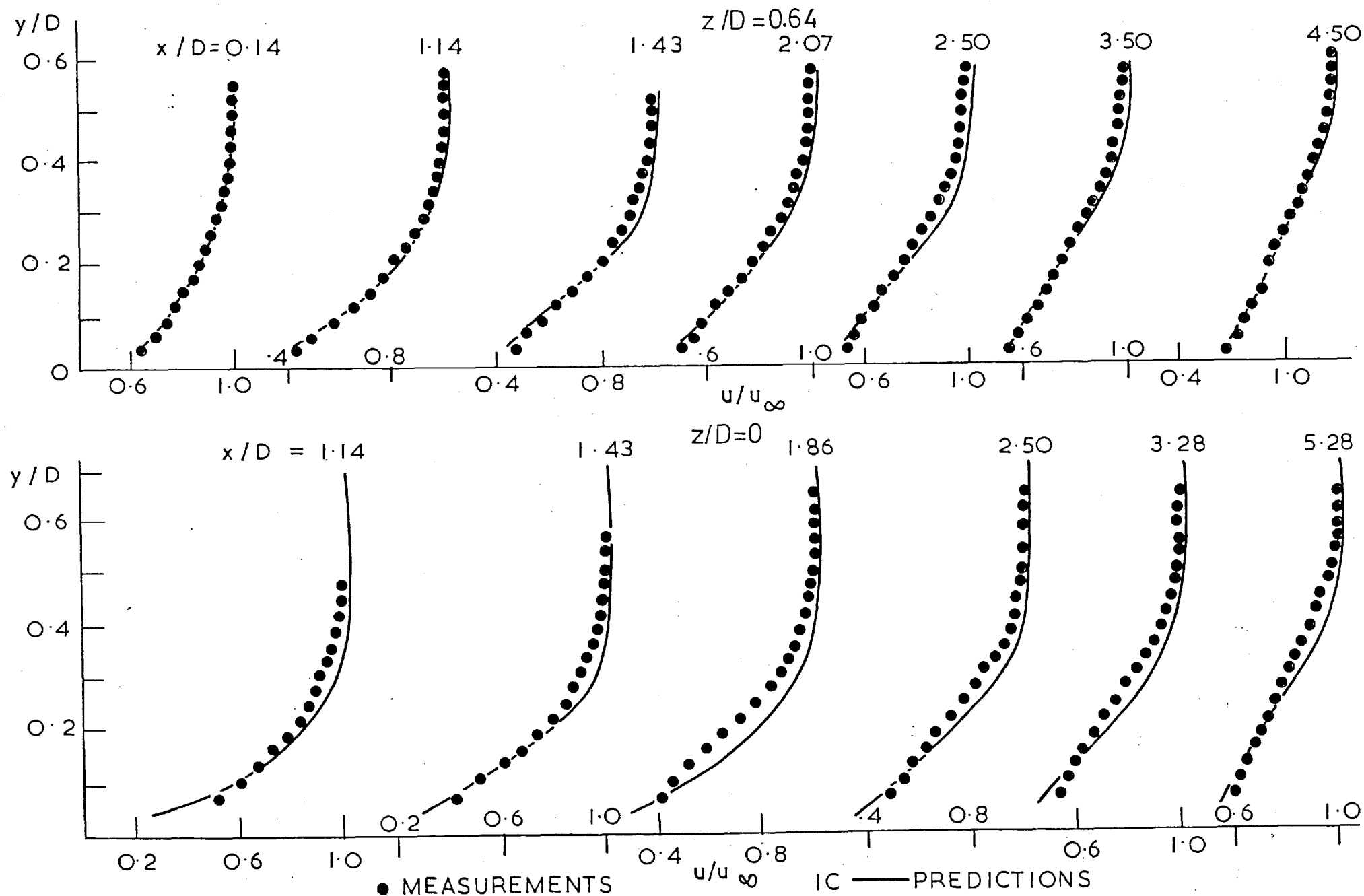


Fig. 62b : Predicted streamwise velocity profiles, comparison with measurements, IC, $\alpha=90^\circ$, $M=0.1$.

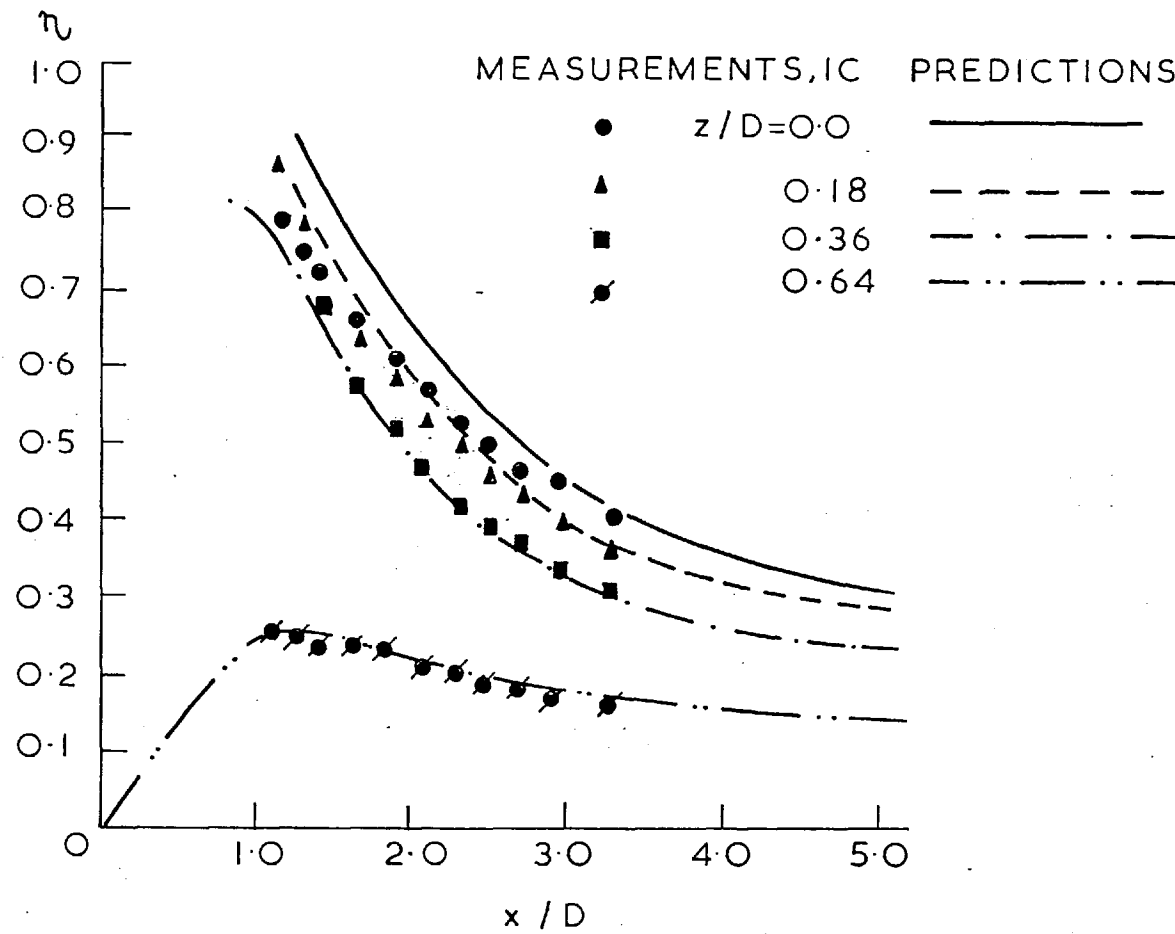


Fig. 62c : Predicted wall film cooling effectiveness,
comparison with measurements IC, $\alpha=90^\circ$,
 $M=0.1$.

— predictions
 ° exper. ref(31)

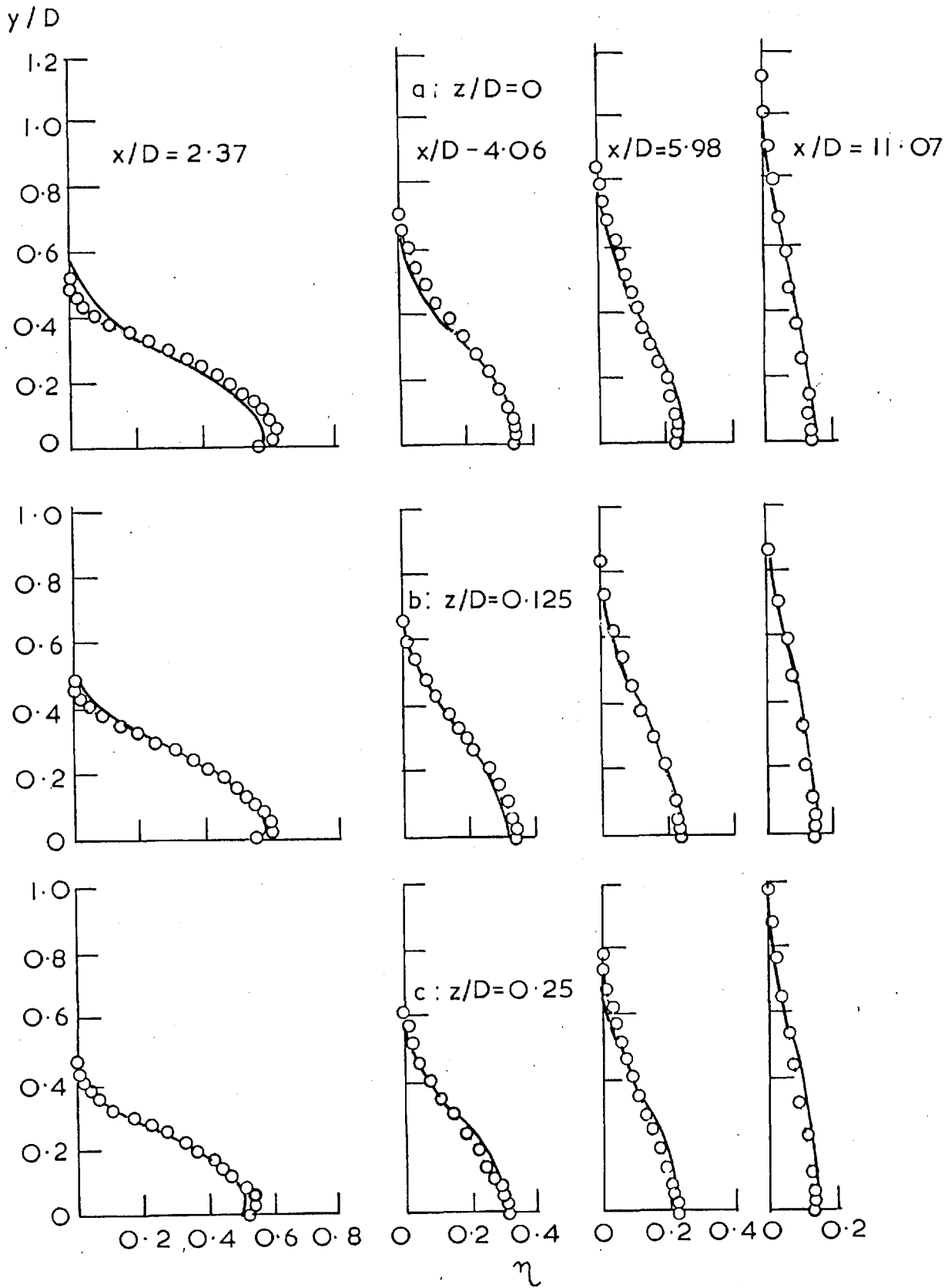


Fig. 63a : Predicted film cooling effectiveness,
 comparison with measurements ref. (31)
 $\alpha = 90^\circ$, $M = 0.1$.

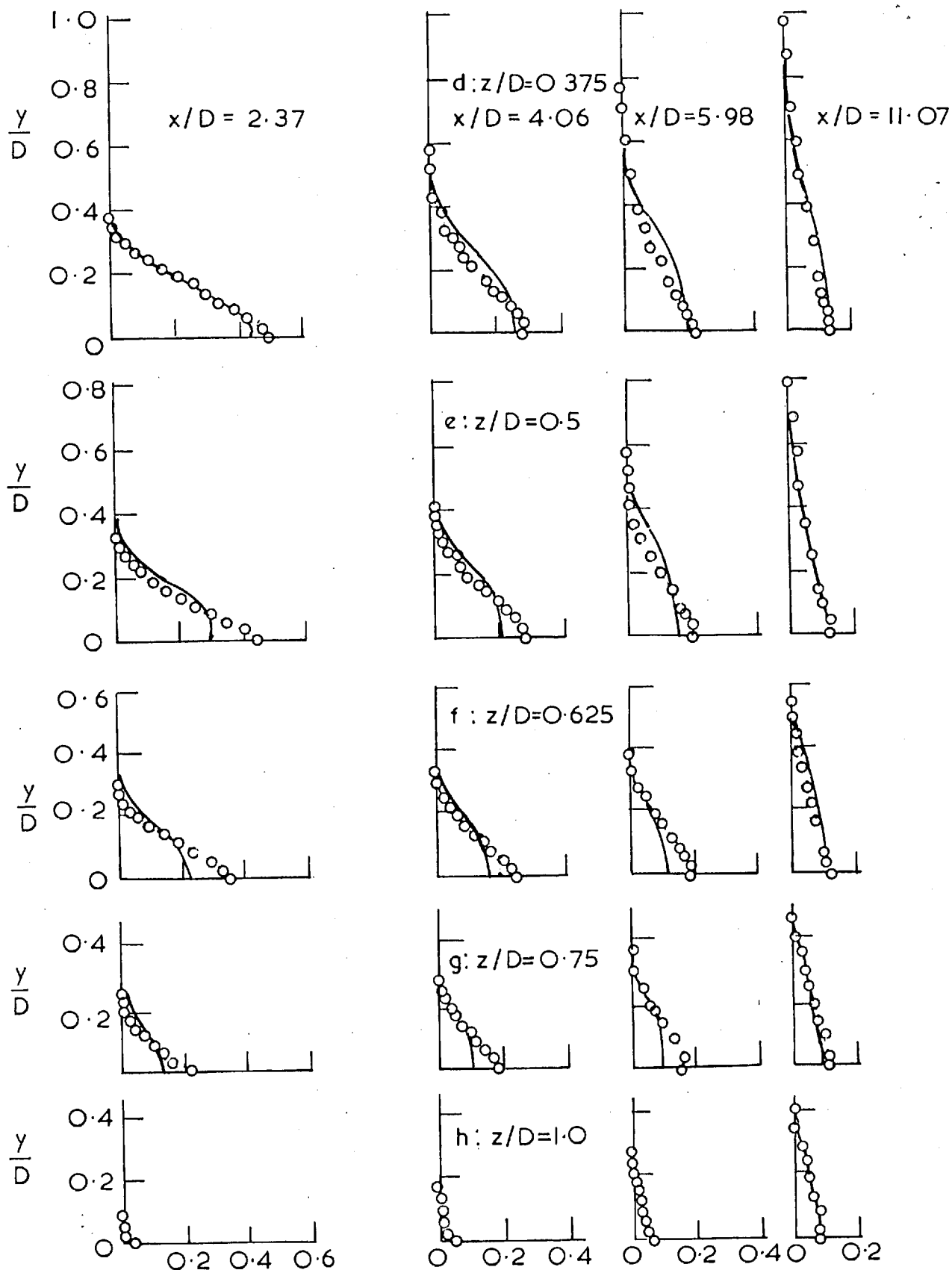


Fig. 63b : Predicted film cooling effectiveness,
comparison with measurements ref. (31)
 $\alpha=90^\circ$, $M=0.1$.

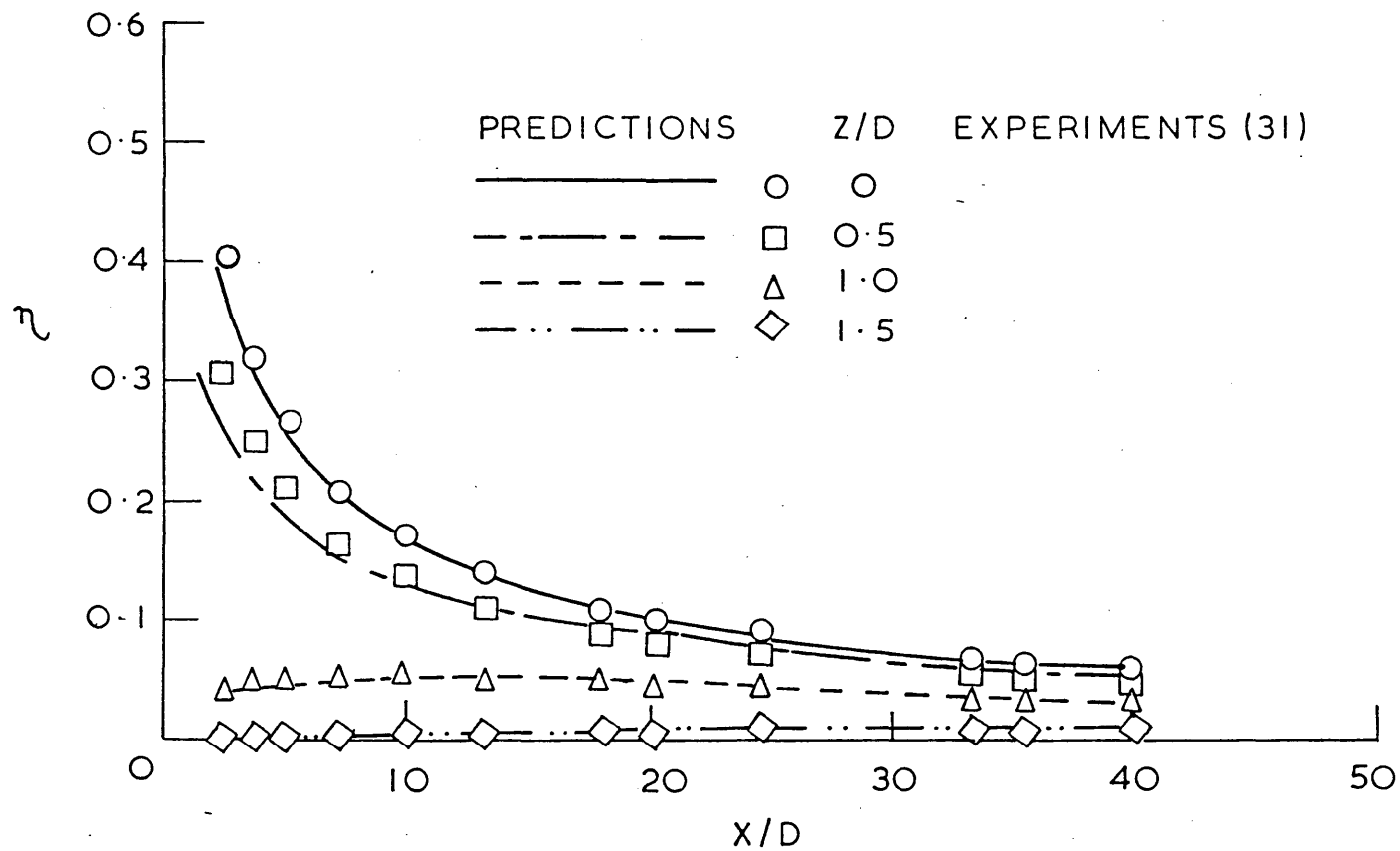
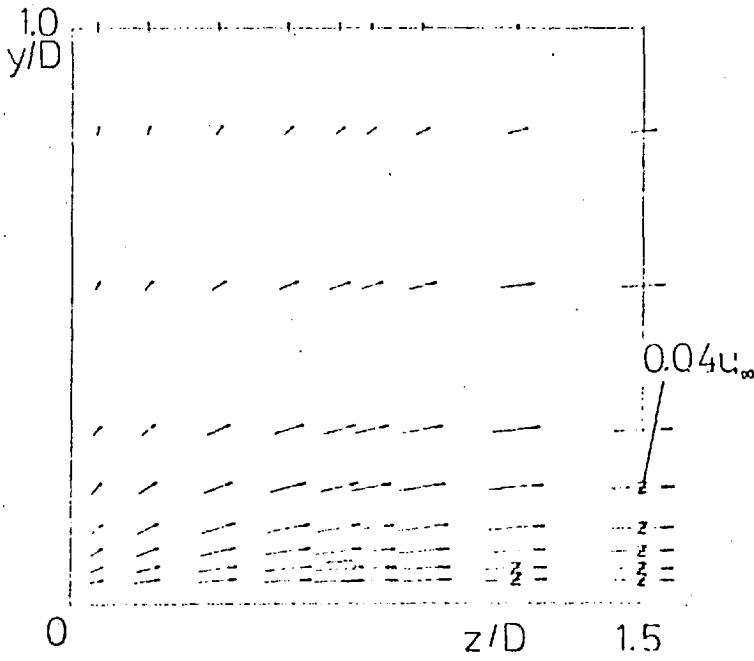
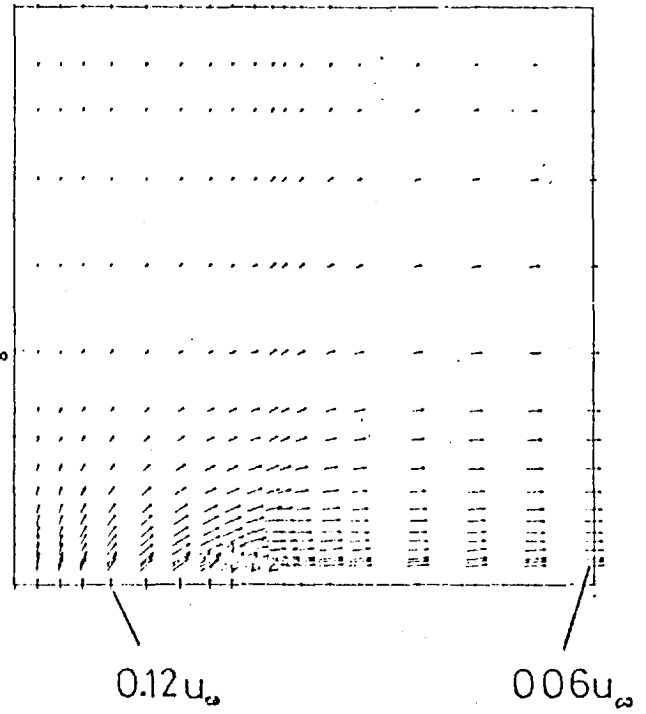


Fig. 63c : Predicted wall film cooling effectiveness,
comparison with measurements ref. (31)
 $\alpha=90^\circ$, $M=0.1$.

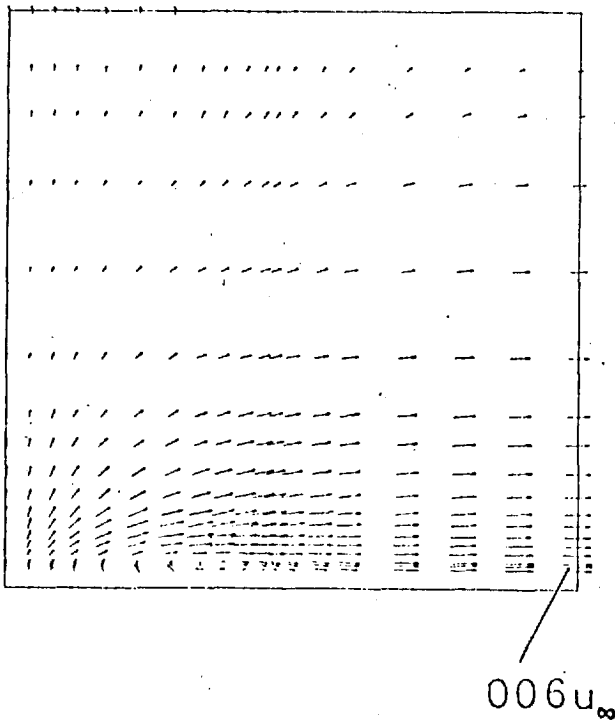
VELOCITY FIELD
 $x/D = -0.2$



VELOCITY FIELD
 $x/D = 0.4$



VELOCITY FIELD
 $x/D = 1.4$



VELOCITY FIELD
 $x/D = 2.0$

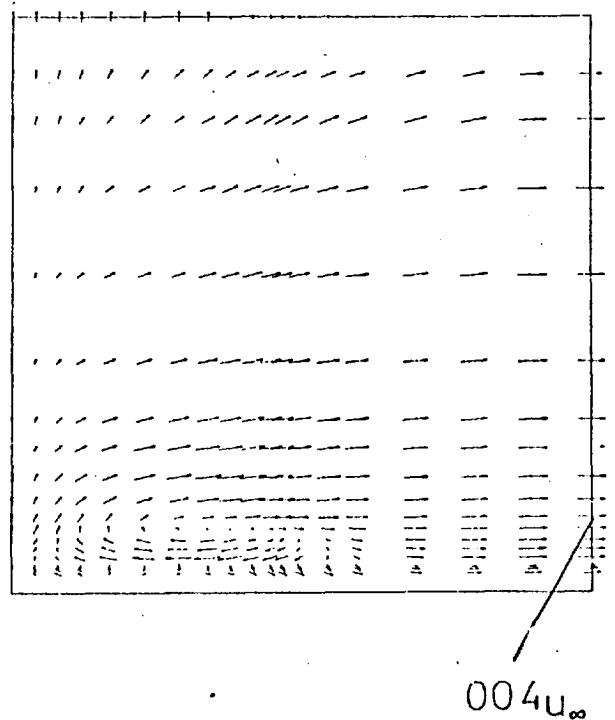
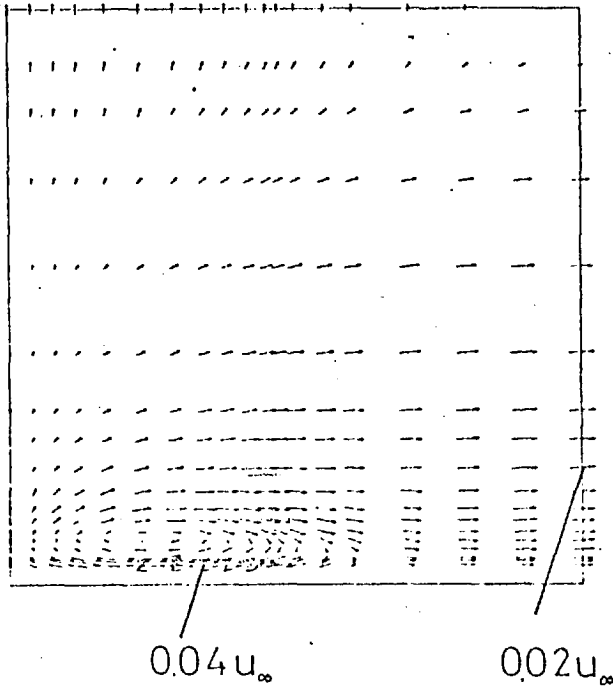
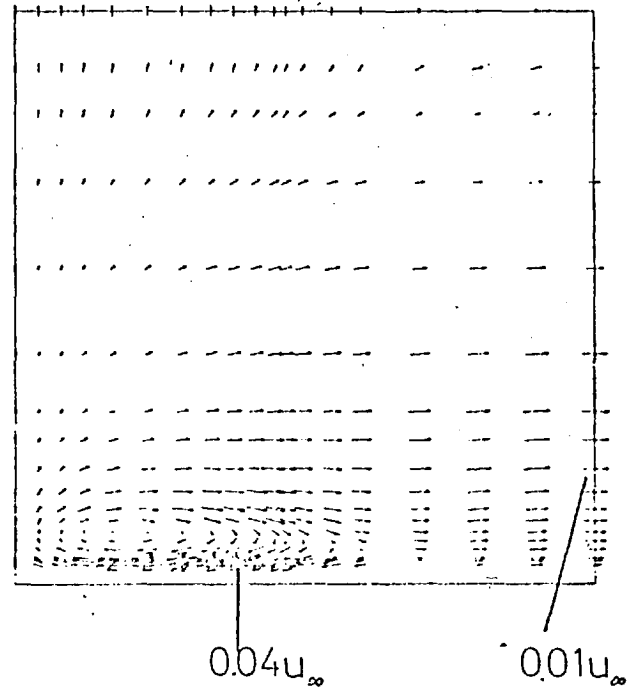


Fig. 64a : Predicted flow patterns on cross-stream
planes, case ref. (31), $\alpha = 90^\circ$, $M = 0.1$.

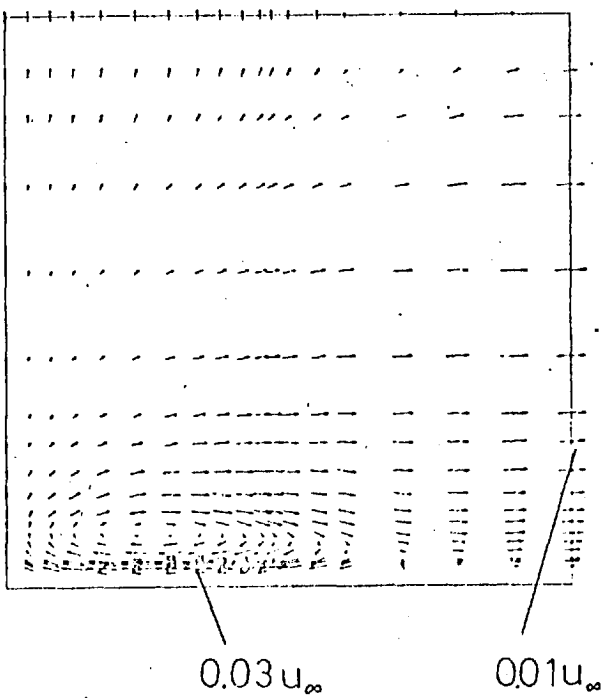
VELOCITY FIELD
 $x/D=2.65$



VELOCITY FIELD
 $x/D=3.40$



VELOCITY FIELD
 $x/D=4.15$



VELOCITY FIELD
 $x/D=5.90$

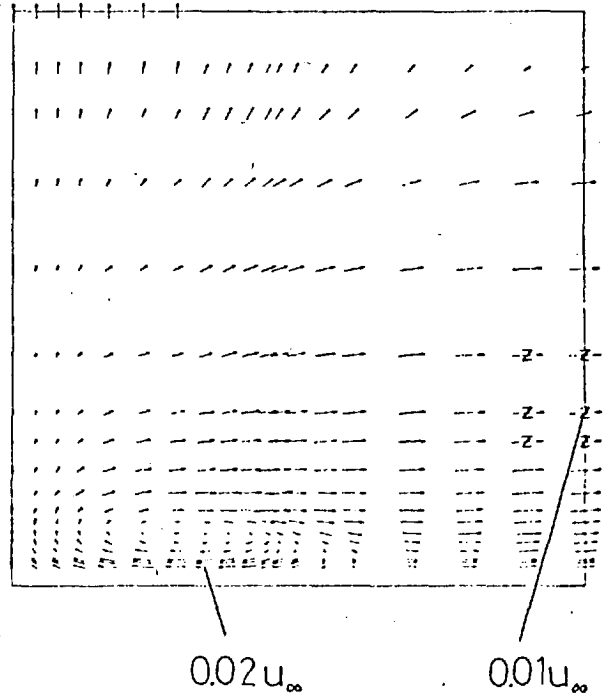


Fig. 64b : Predicted flow pattern on cross-stream
planes, case ref. (31), $\alpha=90^\circ$, $M=0.1$.

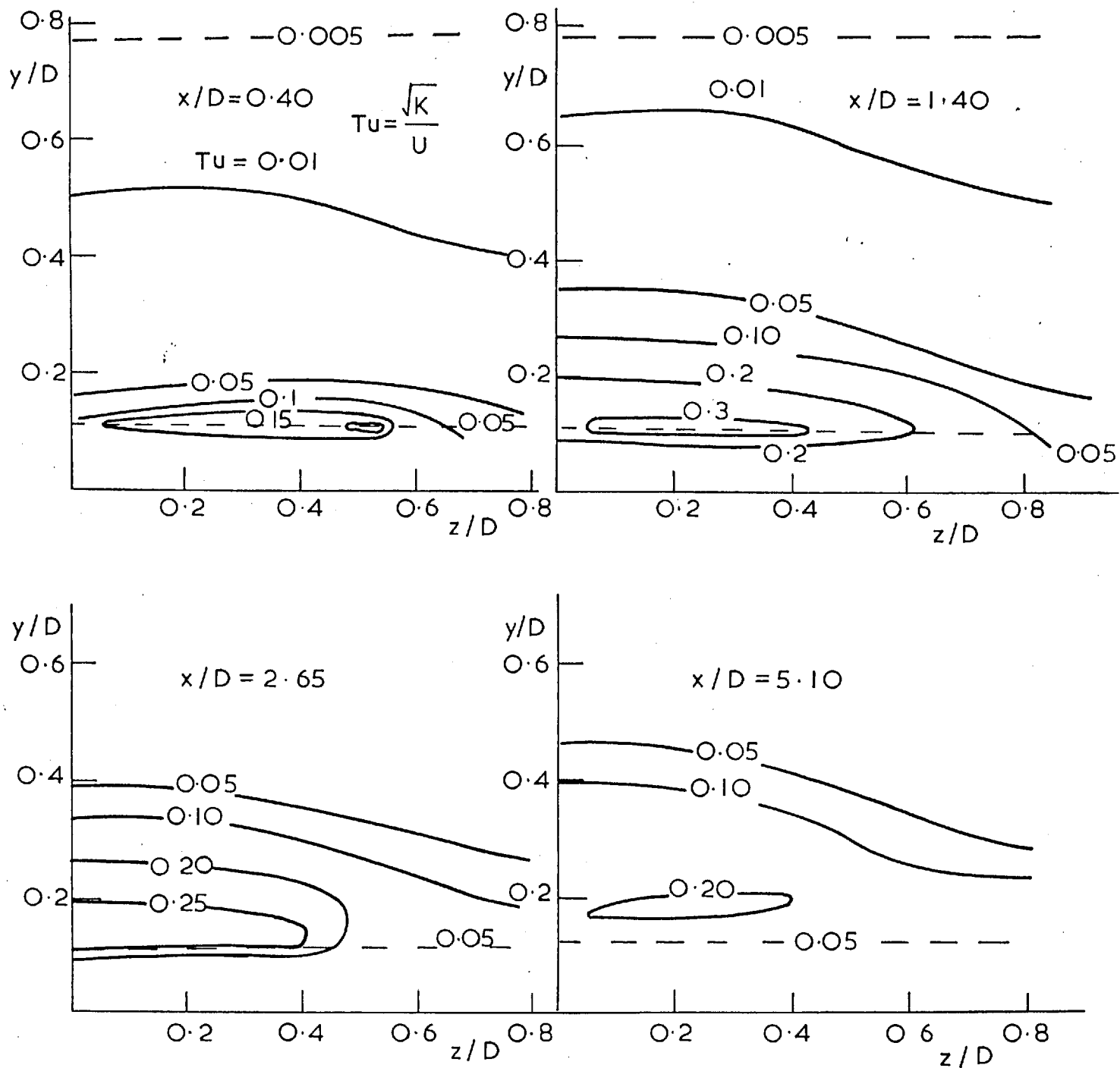


Fig. 65a : Predicted turbulence intensity distribution

on cross-stream planes ref. (31), $\alpha=90^\circ$, $M=0.1$.

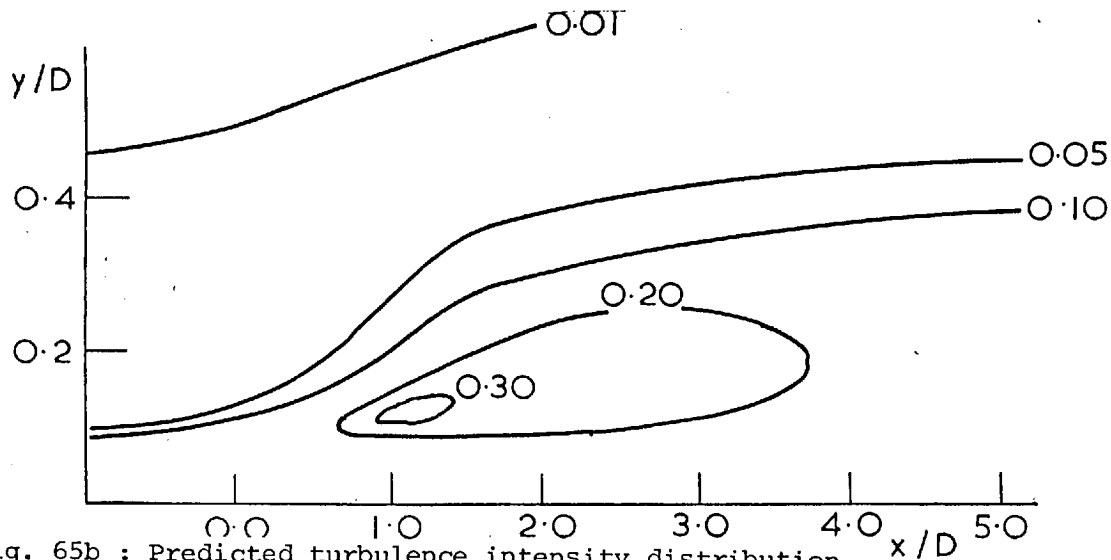


Fig. 65b : Predicted turbulence intensity distribution

on $z/D=0.0$ plane ref. (31), $\alpha=90^\circ$, $M=0.1$.

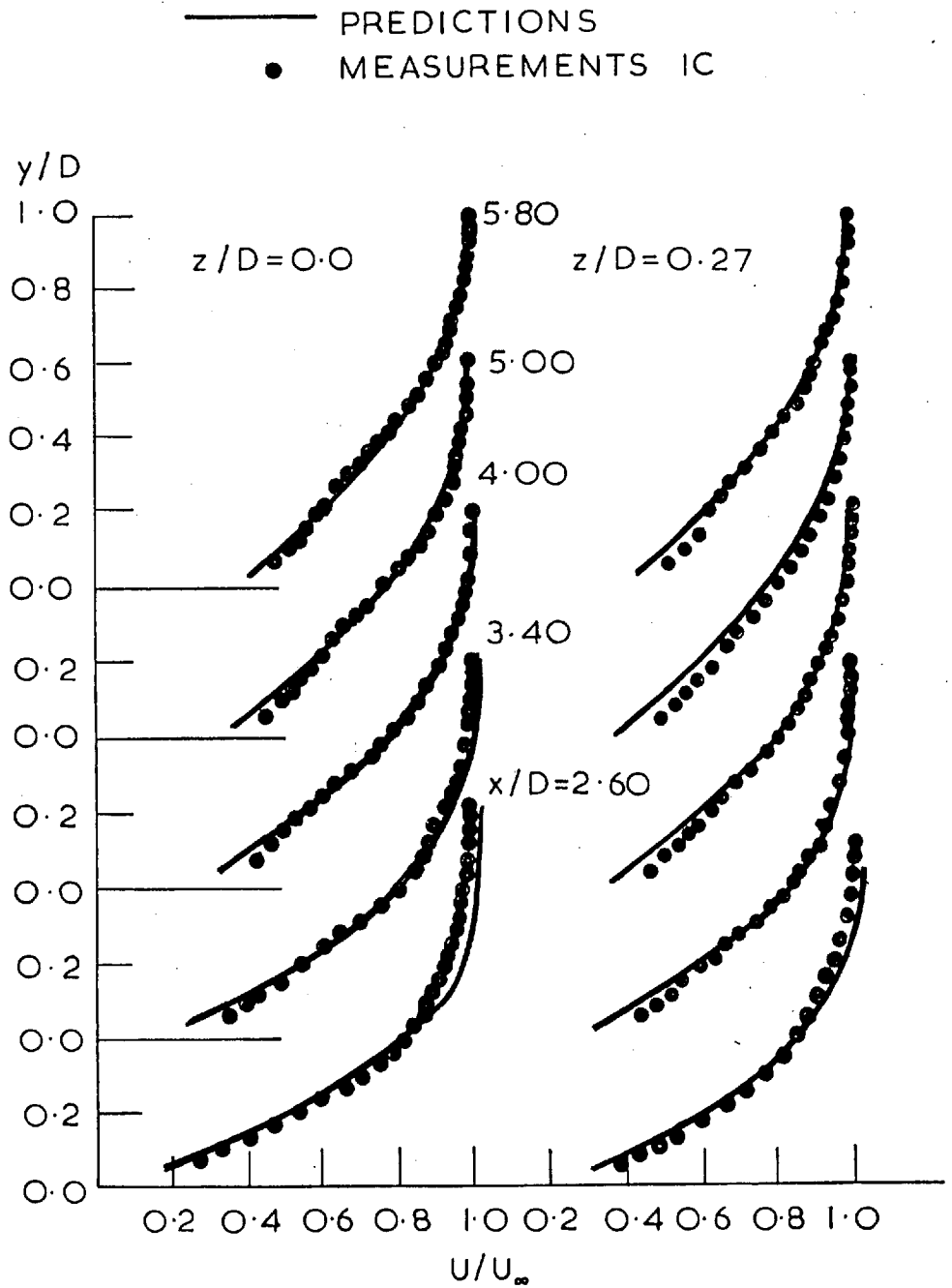


Fig. 66a : Predicted streamwise velocity, comparison
with measurement IC, $\alpha=30^\circ$, $M=0.1$.

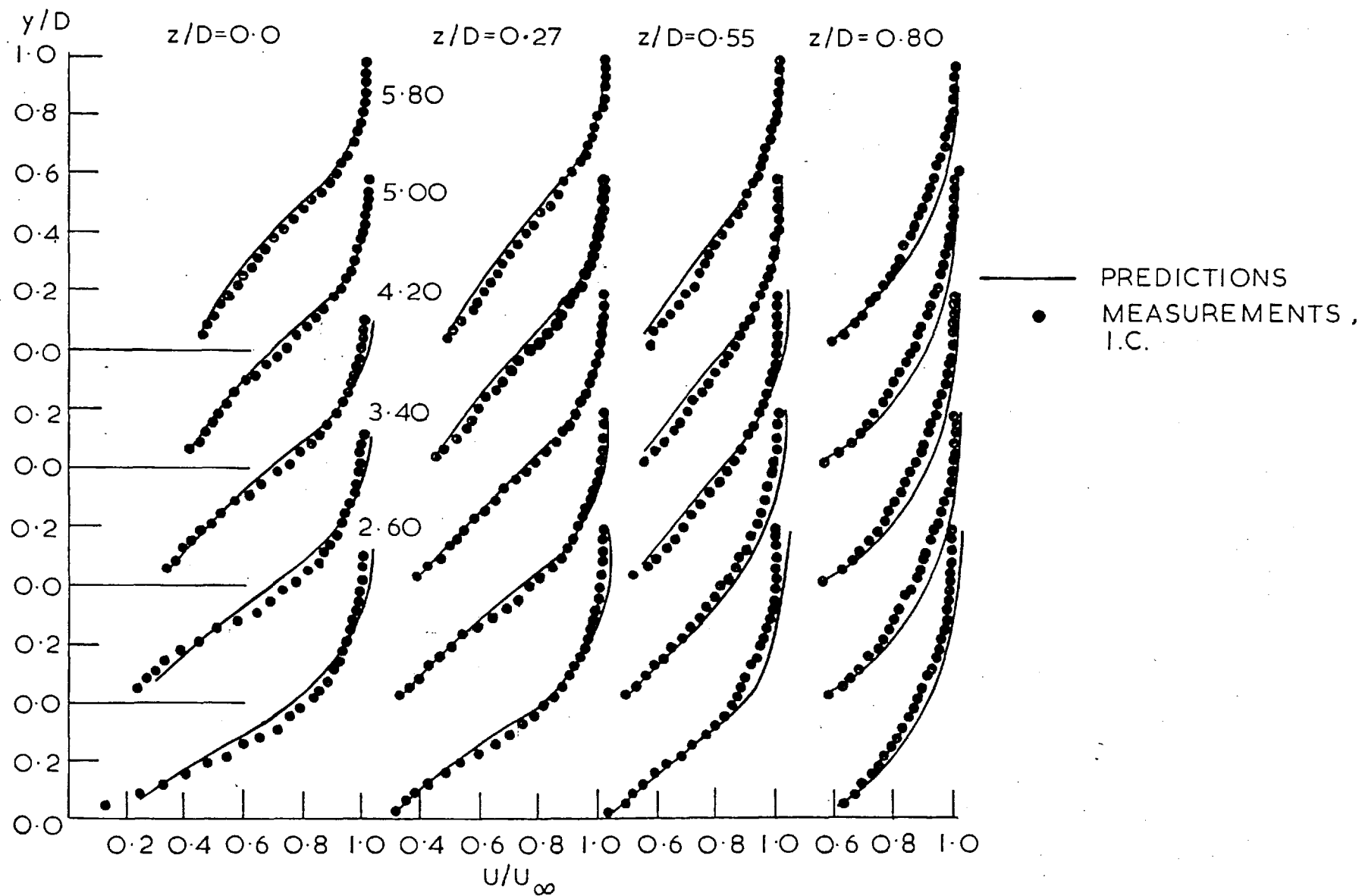


Fig. 66b : Predicted streamwise velocity, comparison
with measurements, IC, $\alpha=30^\circ$, $M=0.2$.

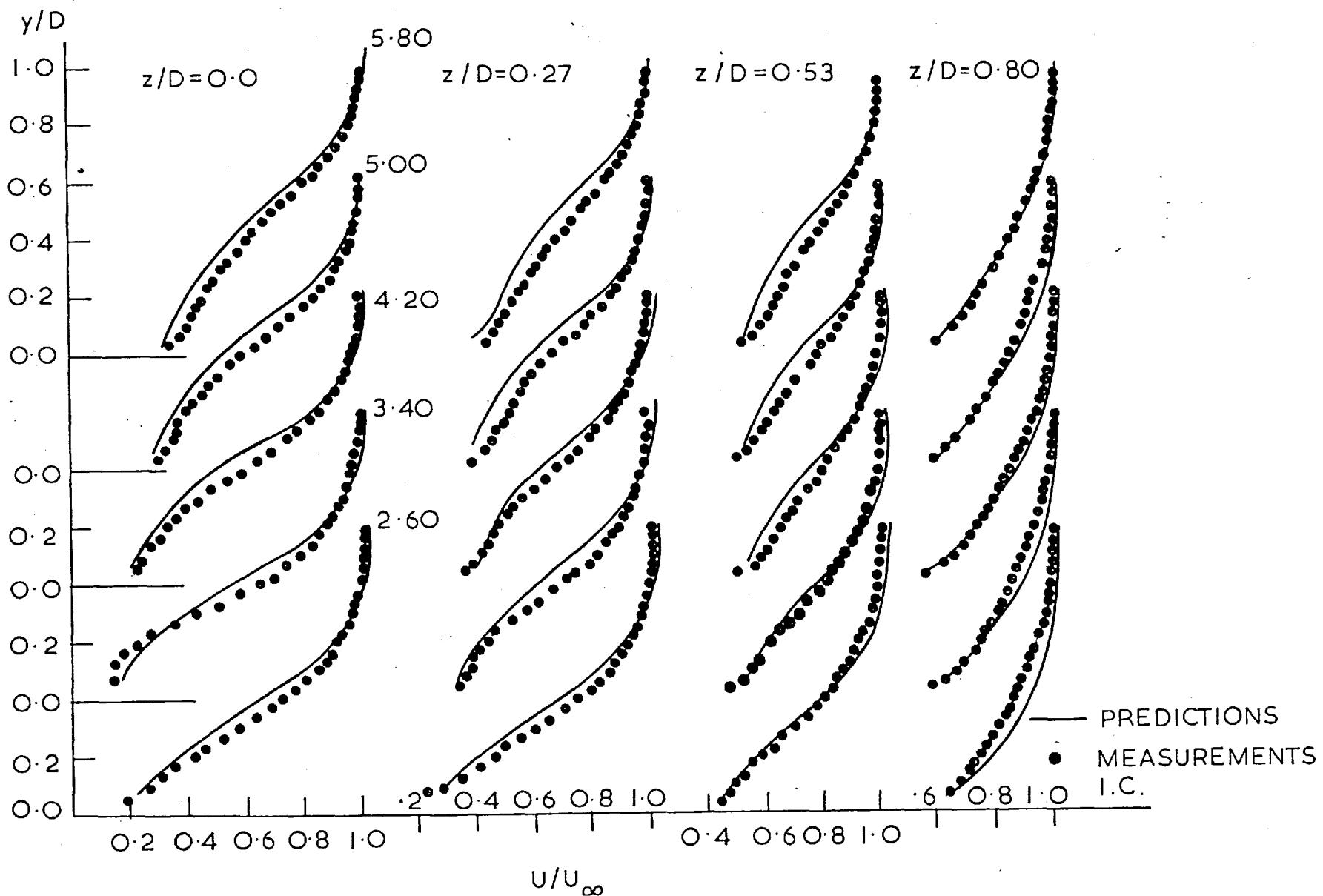


Fig. 66c : Predicted streamwise velocity, comparison

with measurements IC, $\alpha=30^\circ$, $M=0.3$.

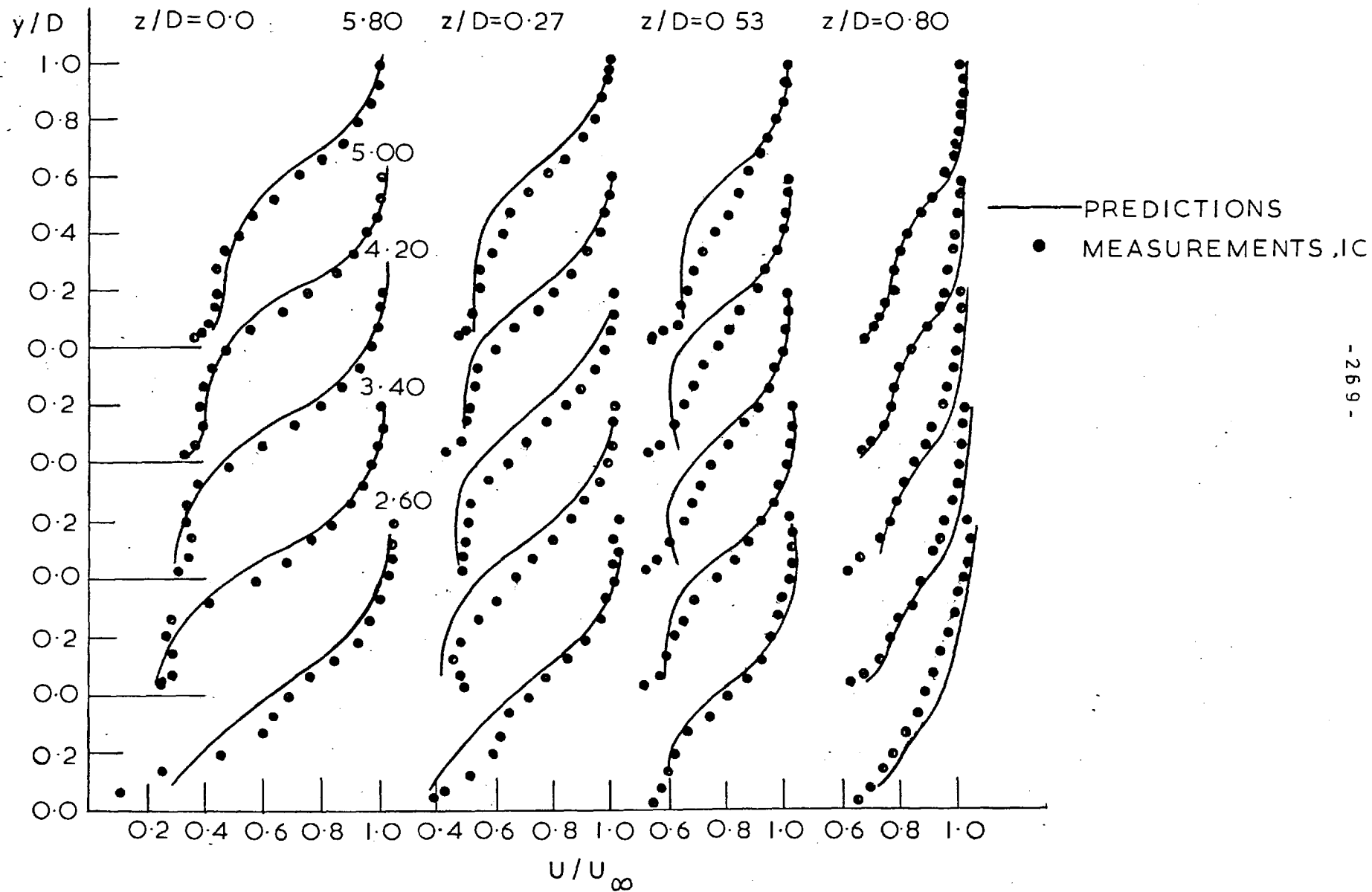


Fig. 66d : Predicted streamwise velocity, comparison

with measurements IC, $\alpha=30^\circ$, $M=0.5$.

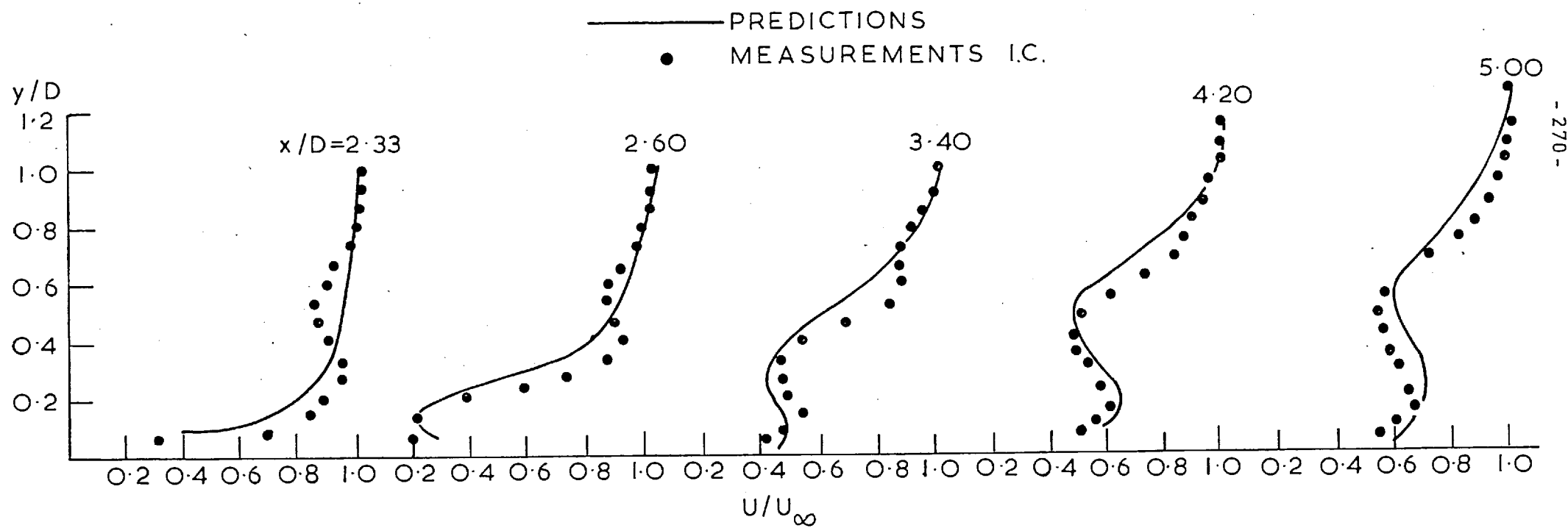


Fig. 66e : Predicted streamwise velocity, comparison
 with measurements IC, $\alpha=30^\circ$, $M=1.0$.

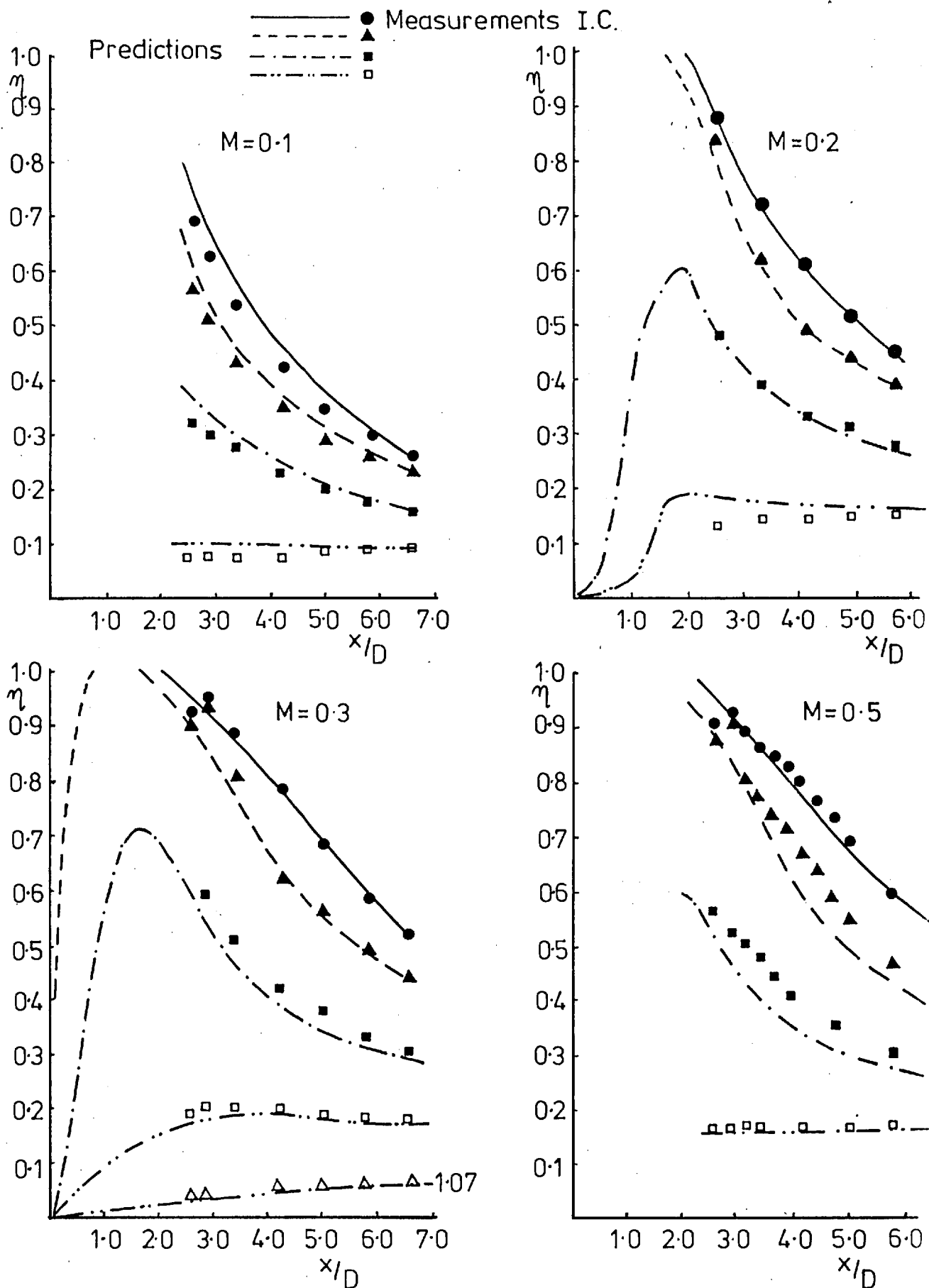


Fig. 67a : Predicted wall film cooling effectiveness,
comparison with measurements IC, $\alpha=30^\circ$,
 $M=0.1, 0.2, 0.3, 0.5$.

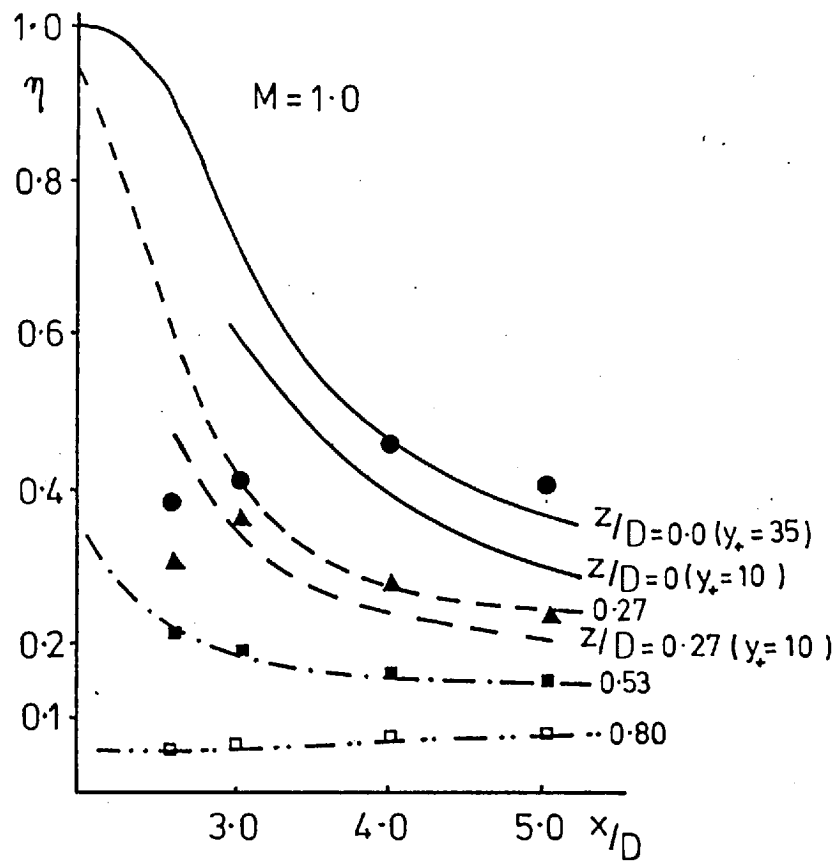


Fig. 67b : Predicted wall film cooling effectiveness,
comparison with measurements IC, $\alpha=30^\circ$,
 $M=1.0$.

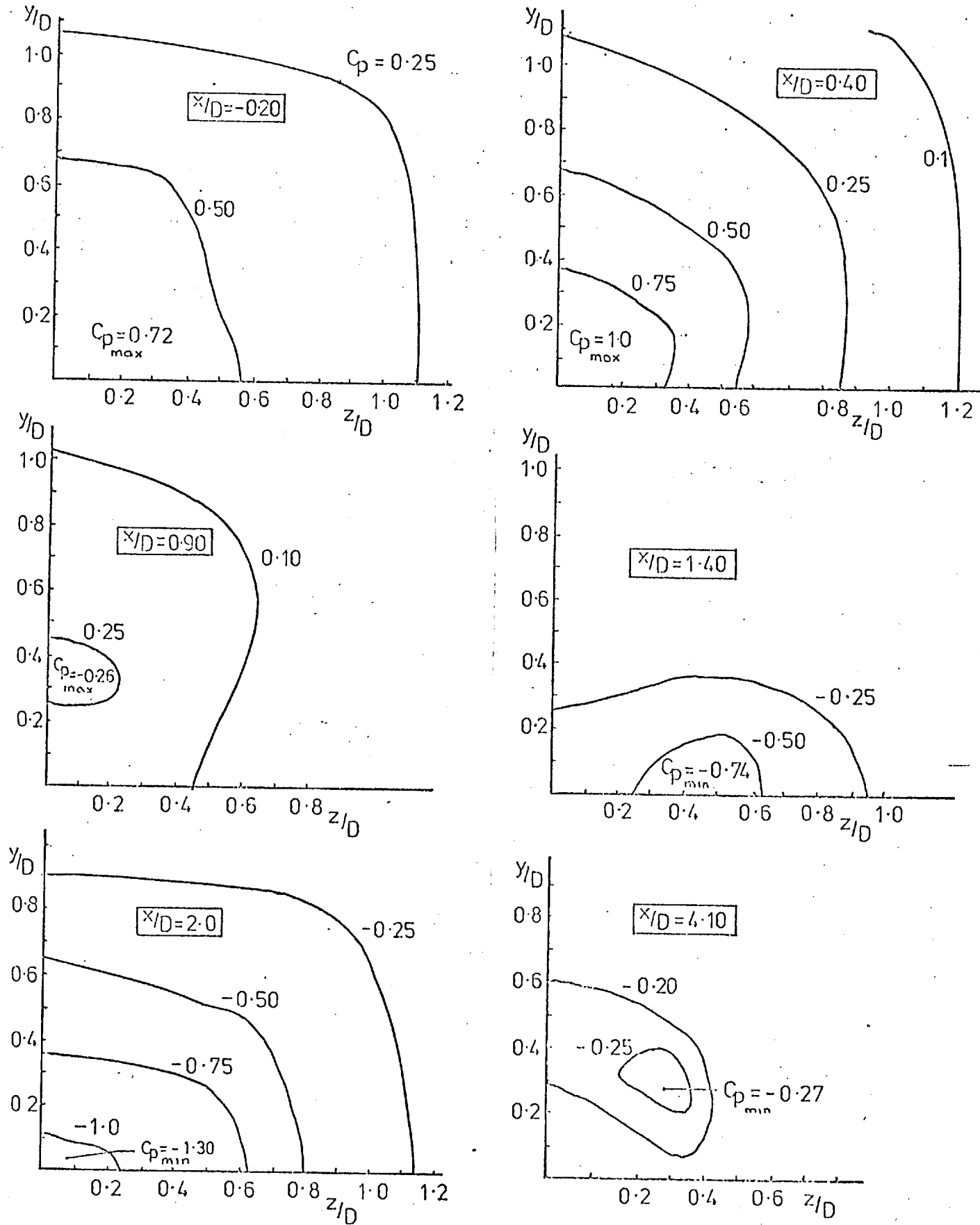
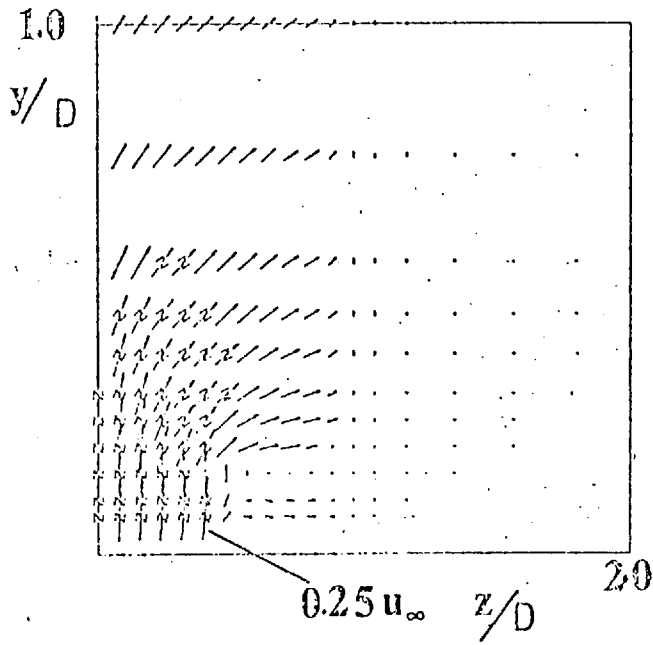
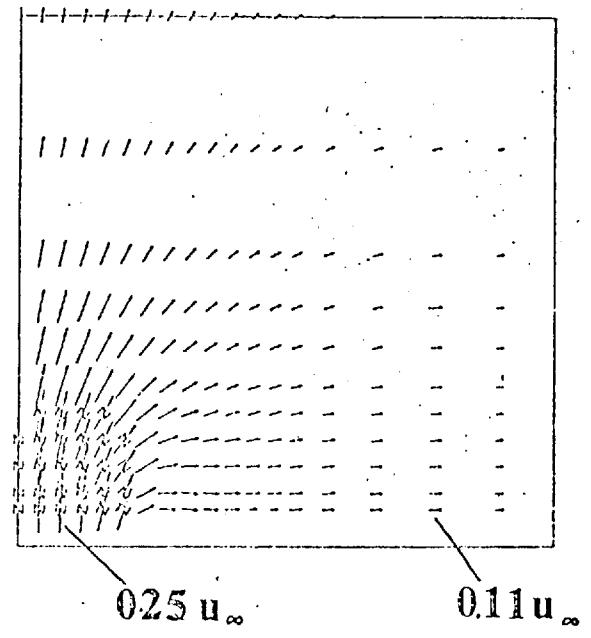


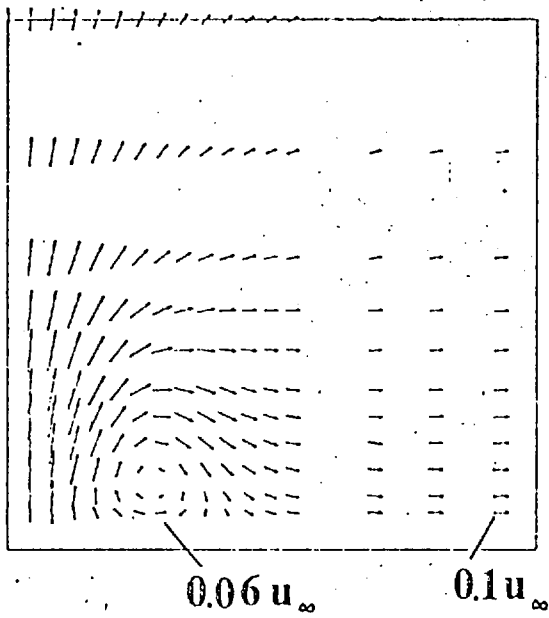
Fig. 68 : Predicted pressure distribution on cross-stream planes IC, $\alpha=30^\circ$, $M=0.5$.



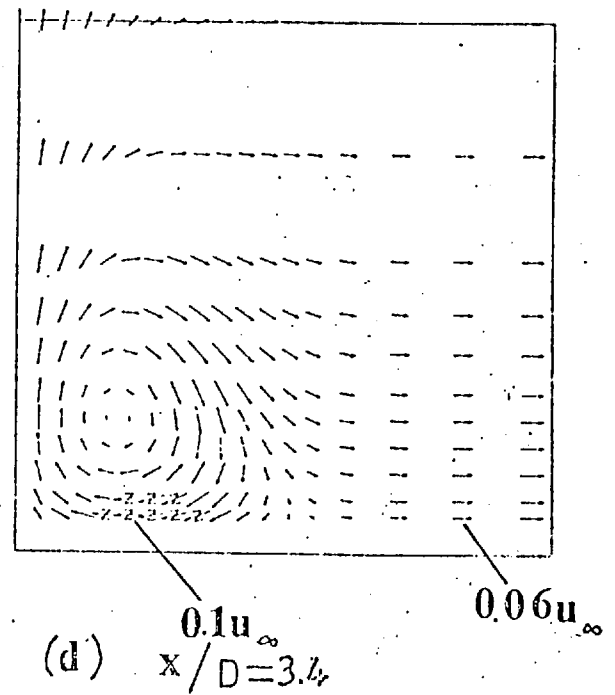
(a) $x/D = 0.9$



(b) $x/D = 1.4$



(c) $x/D = 2.0$



(d) $x/D = 3.4$

Fig. 69 : Predicted flow pattern on cross-stream
planes, IC, $\alpha = 30^\circ$, $M = 0.5$.

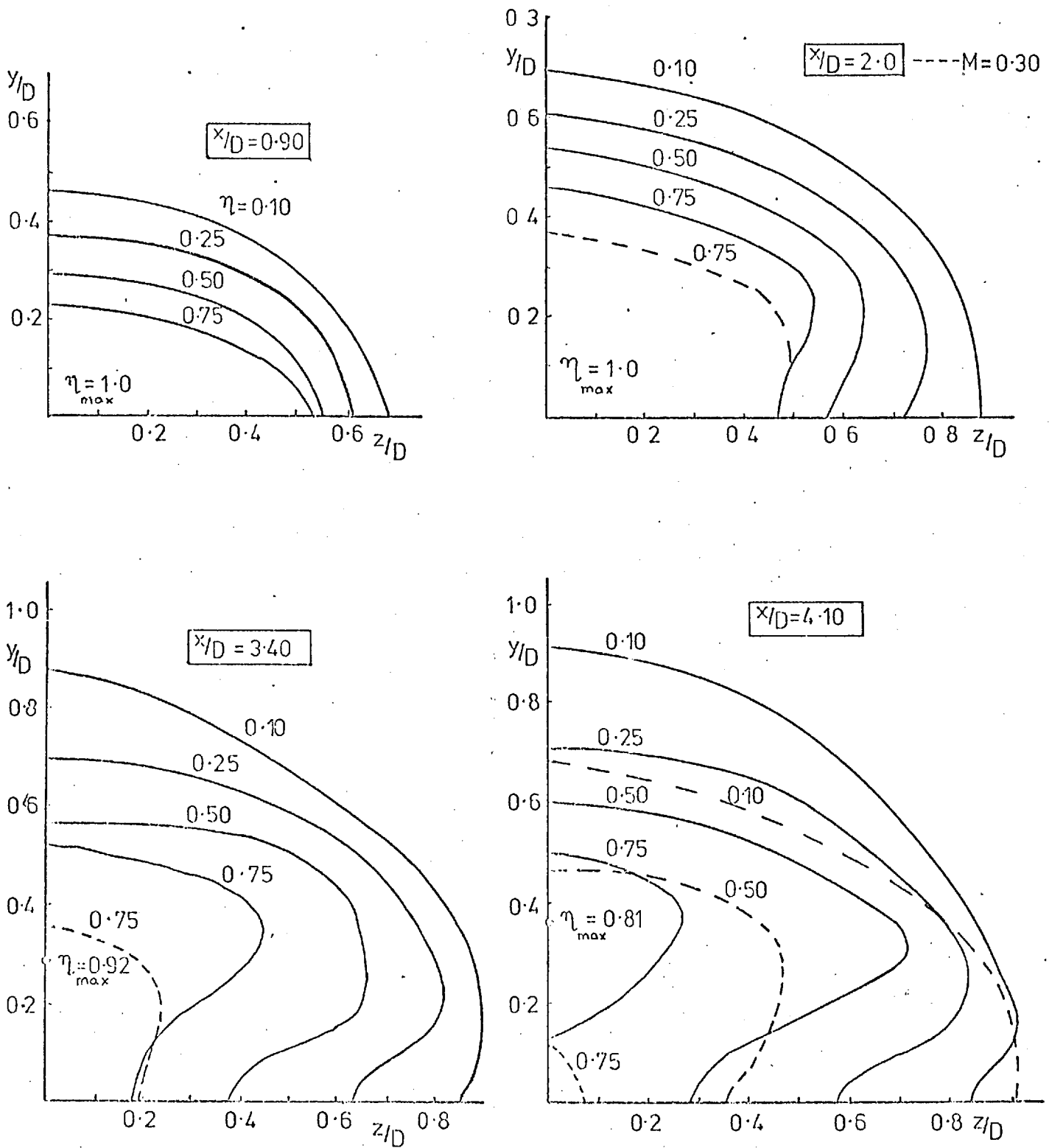


Fig. 70 : Predicted effectiveness contours on cross-stream planes IC, $\alpha = 30^\circ$, $M = 0.5$.

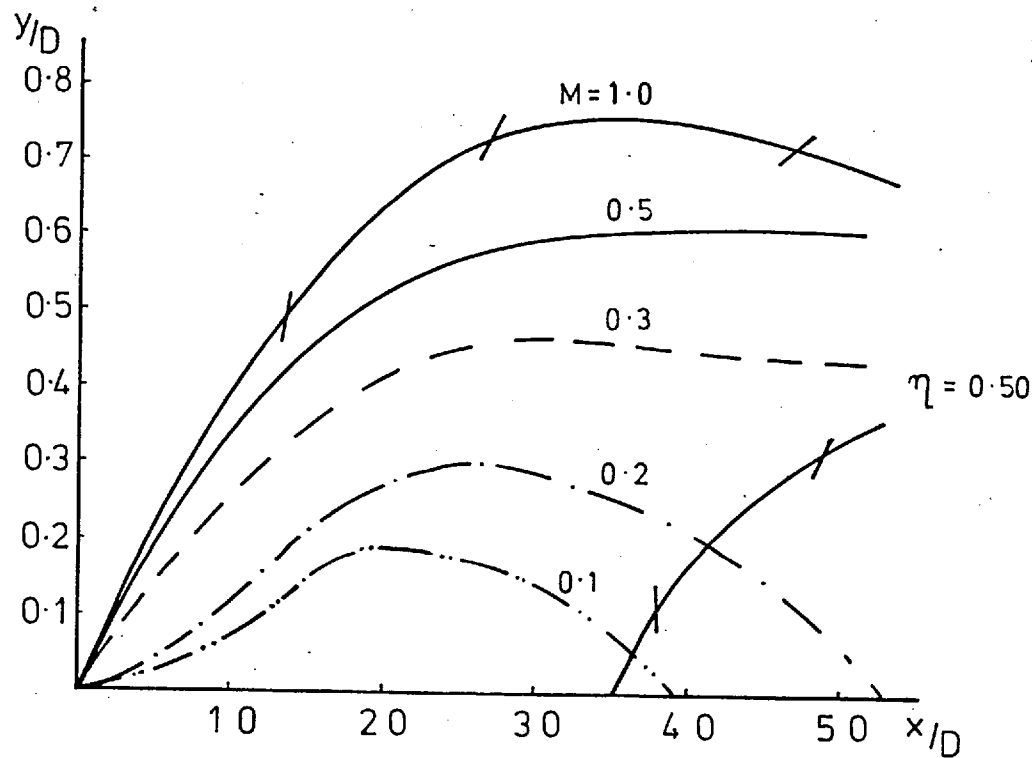


Fig. 7la : Predicted location of $\eta=50\%$ contour on $z/D=0$ plane IC, $\alpha=30^\circ$.

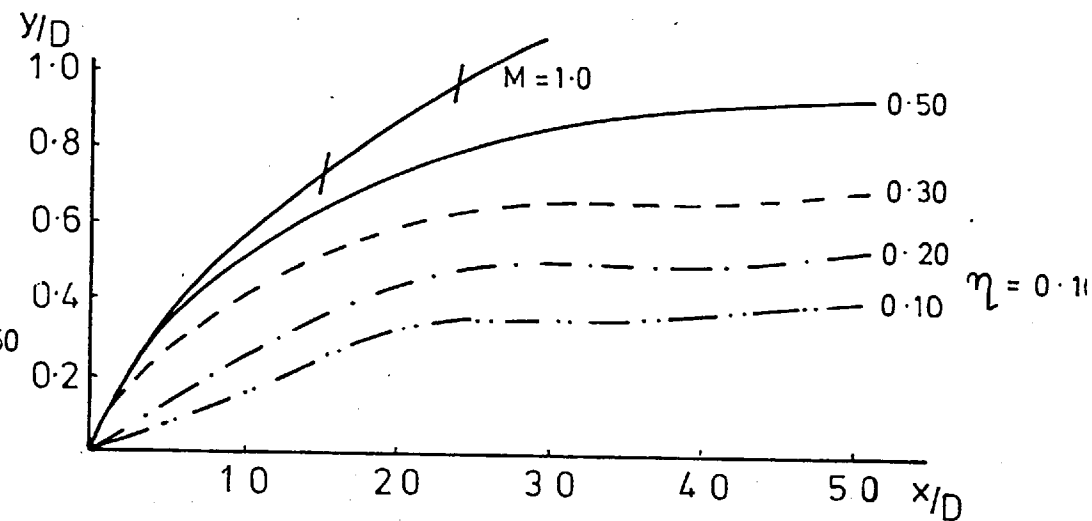


Fig. 7lb : Predicted location of $\eta=10\%$ contour on $z/D=0.0$, plane IC, $\alpha=30^\circ$.

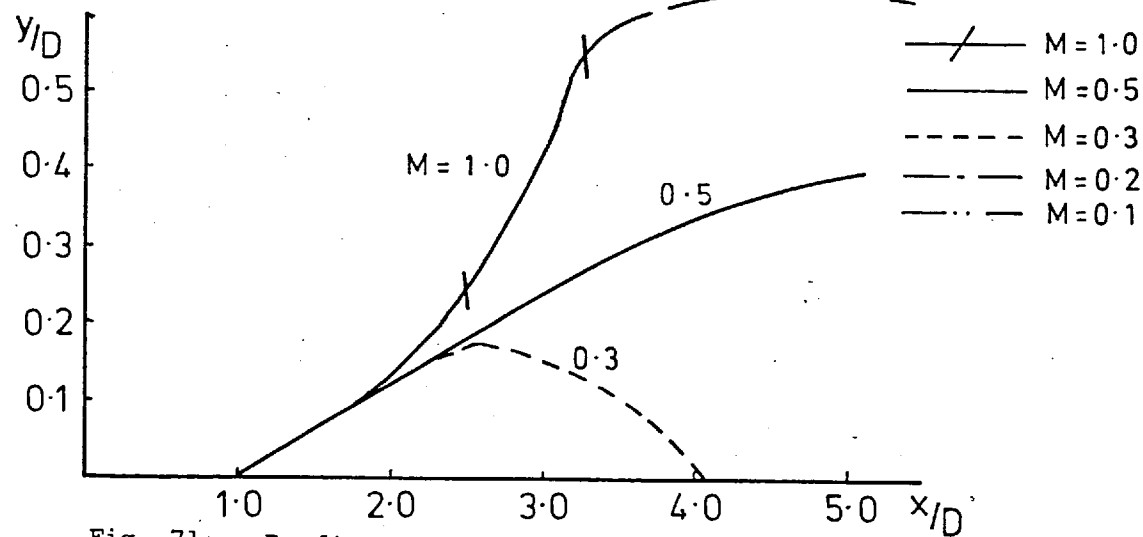


Fig. 7lc : Predicted location of $\eta_{\max}$ contour on $z/D=0.0$ plane IC, $\alpha=30^\circ$.

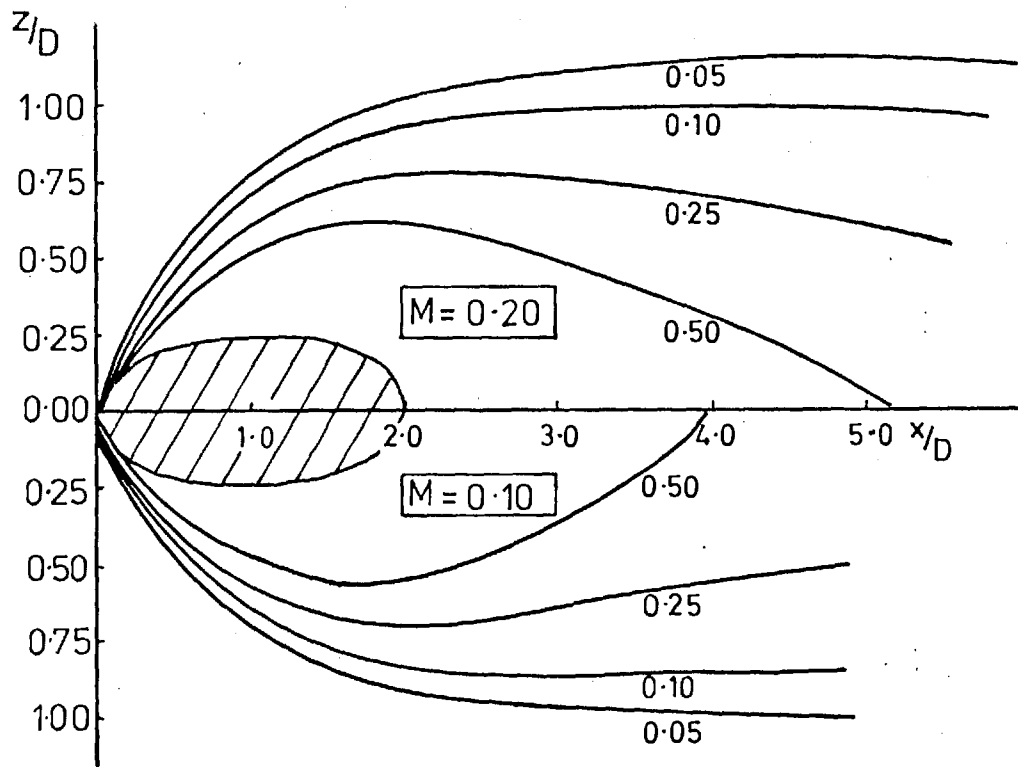


Fig. 72a : Predicted wall film cooling effectiveness IC , $\alpha=30^\circ$

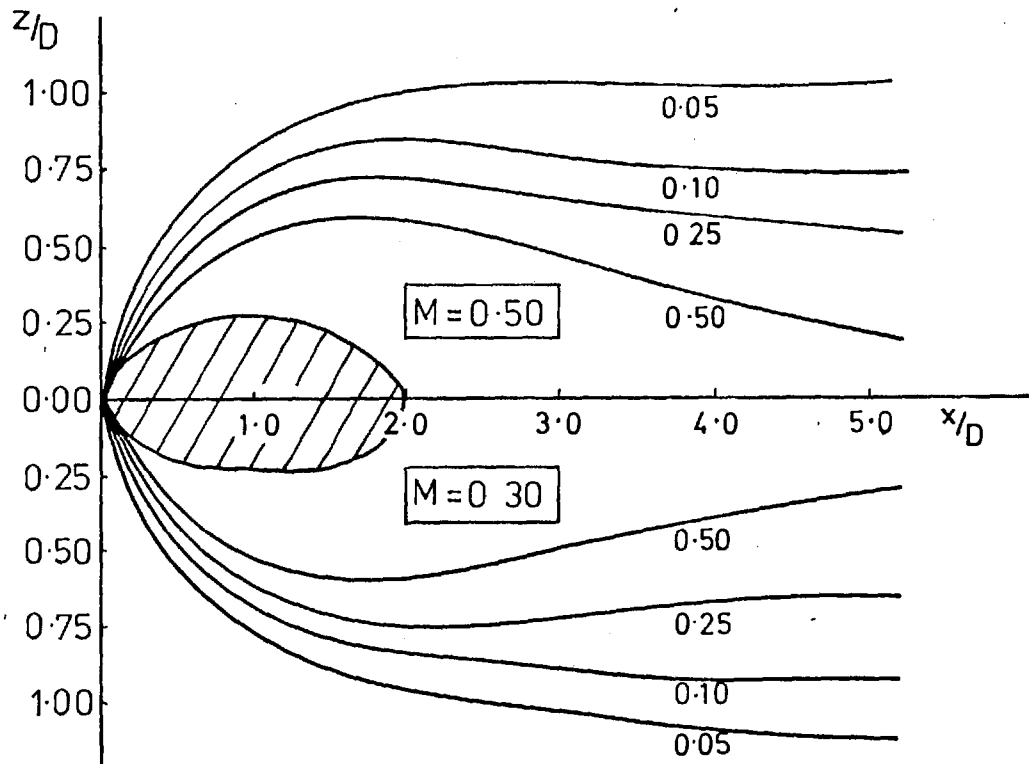


Fig. 72b : Predicted wall film cooling effectiveness IC , $\alpha=30^\circ$

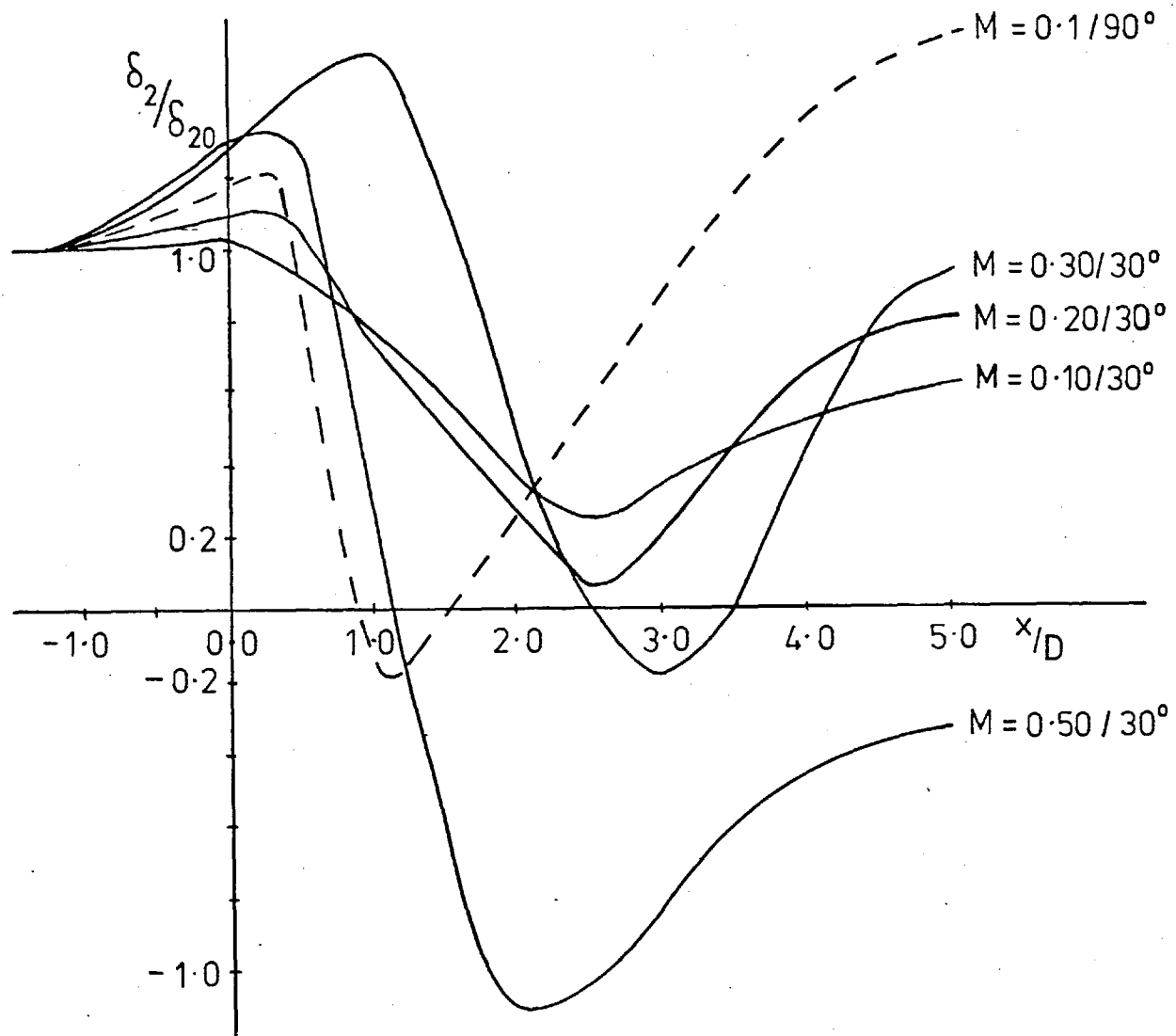


Fig. 73 : Predicted momentum losses for various
blowing rates on $z/D=0$, plane IC.

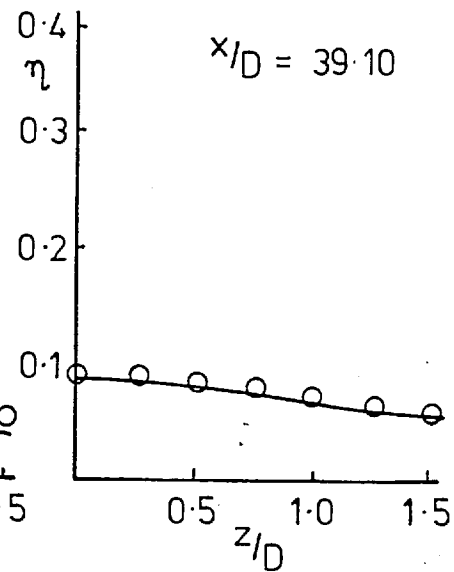
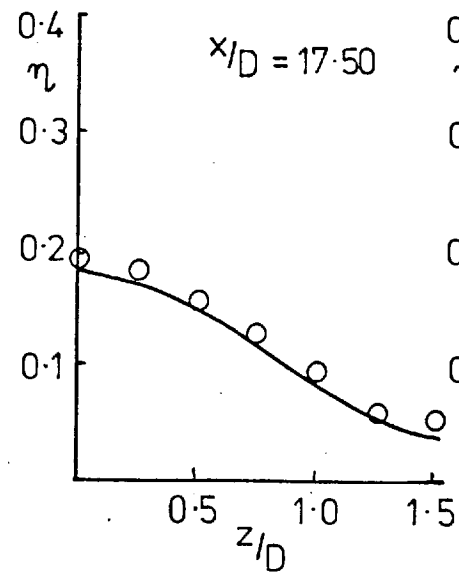
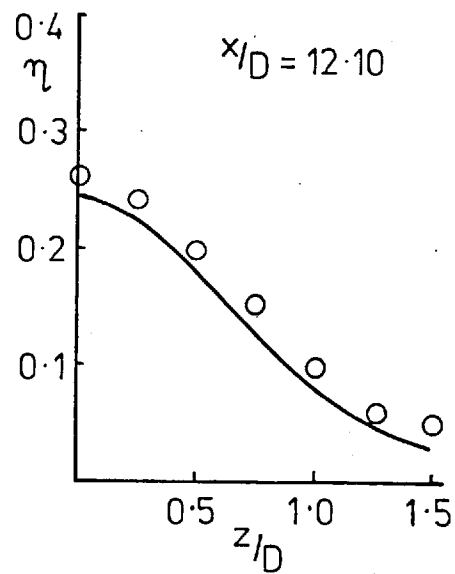
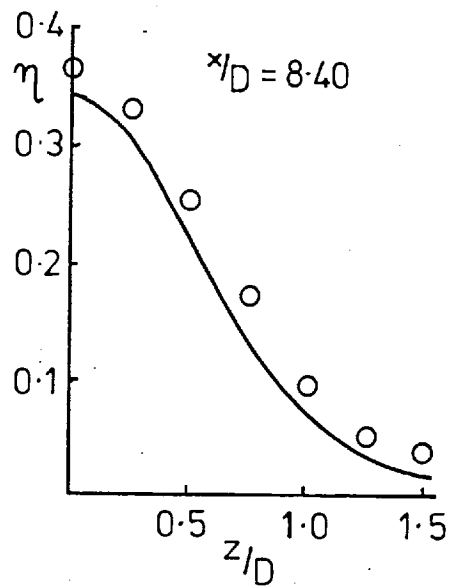
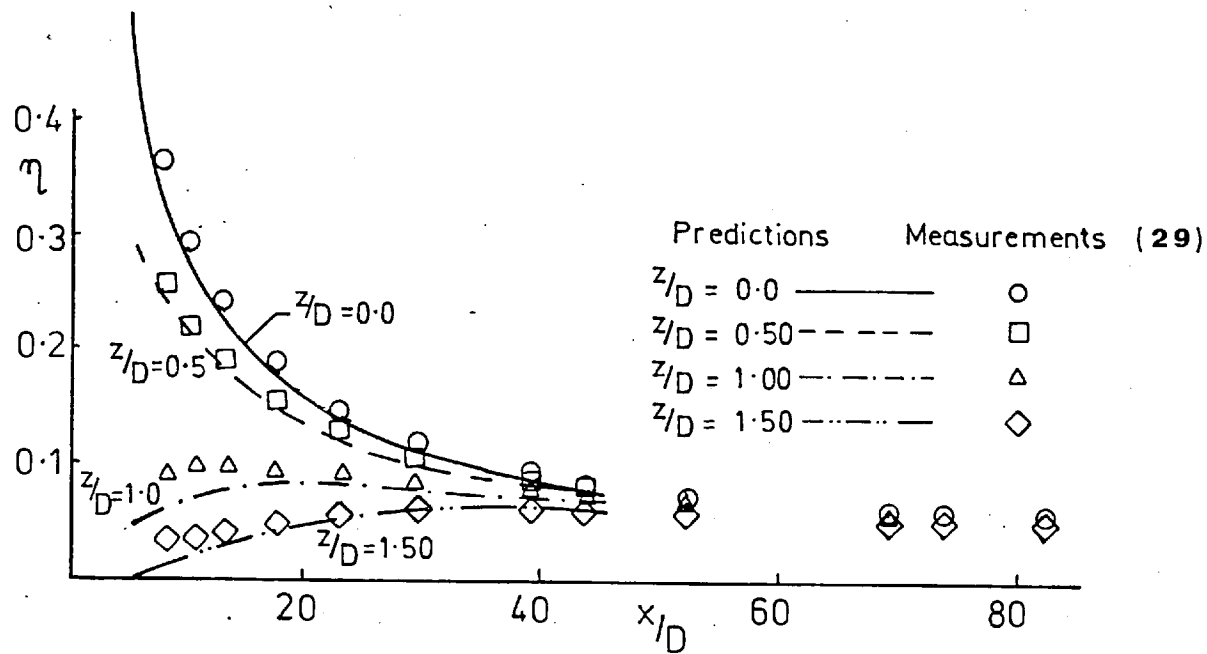


Fig. 74 : Predicted wall film cooling effectiveness,
 comparison with measurements ref. (29),
 $\alpha = 35^\circ$, $M = 0.2$.

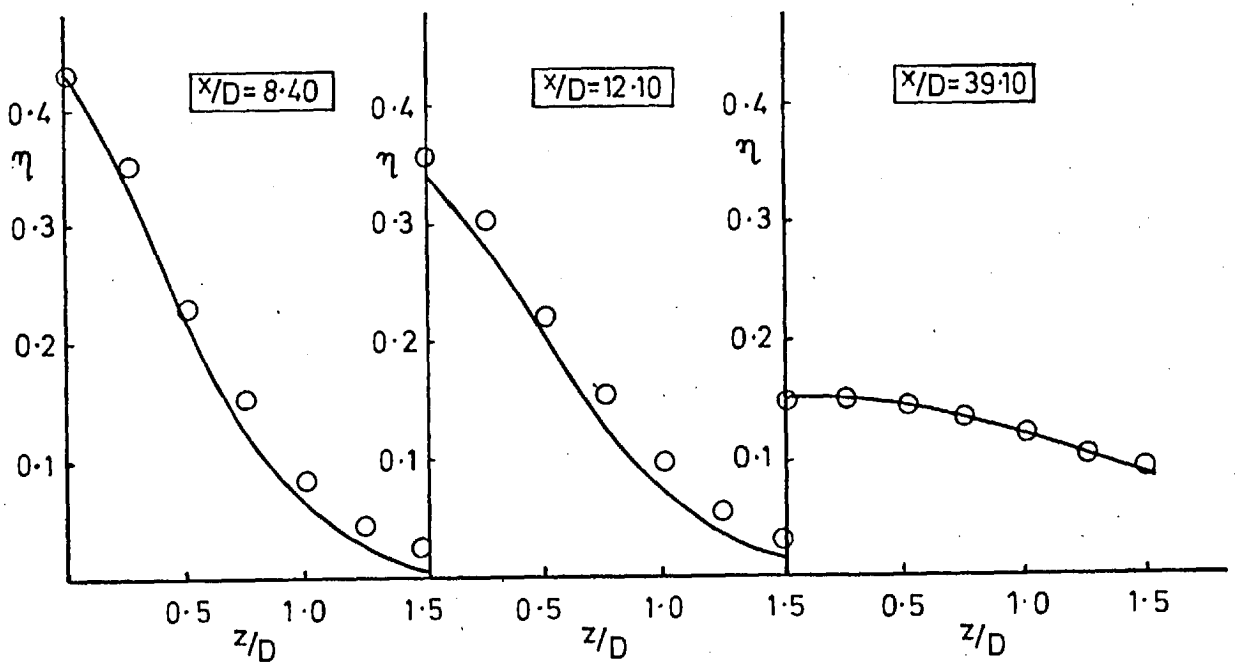
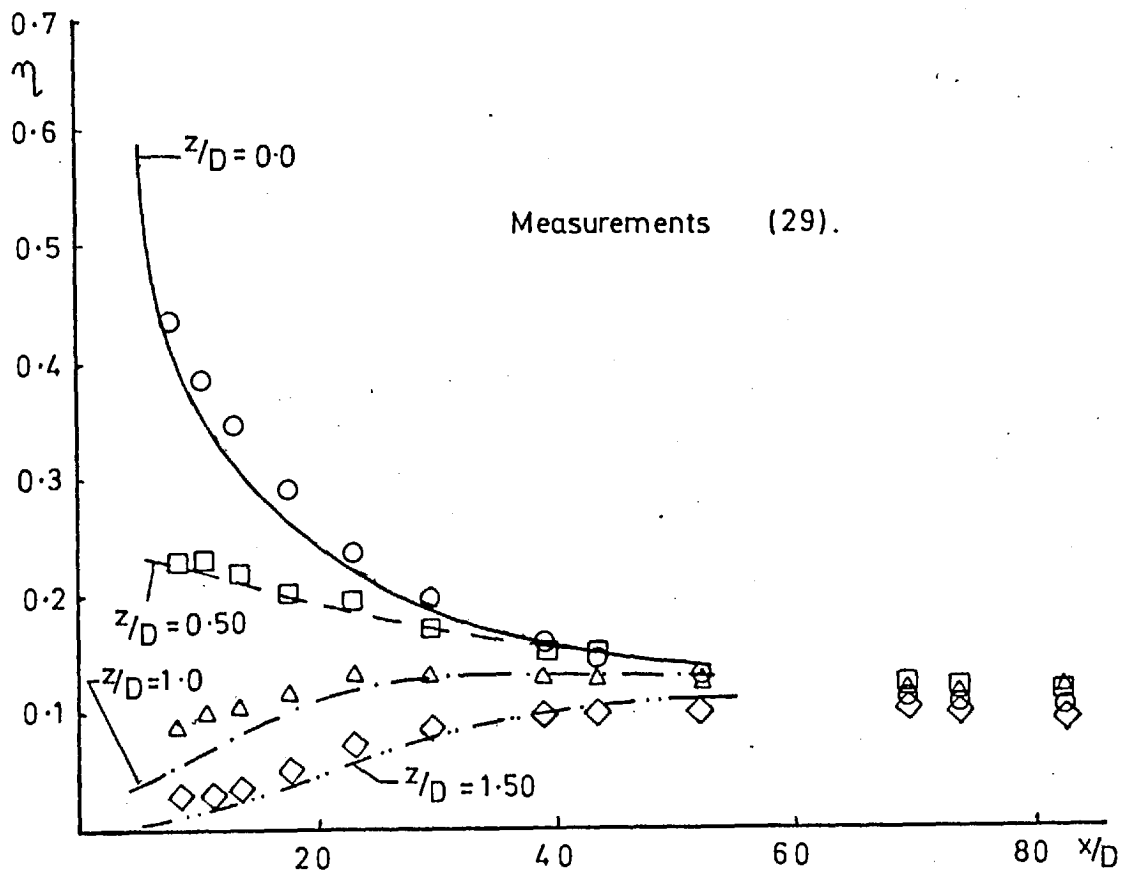


Fig. 75 : Predicted wall film cooling effectiveness,
comparison with measurements ref. (29),
 $\alpha = 35^\circ$, $M = 0.5$.

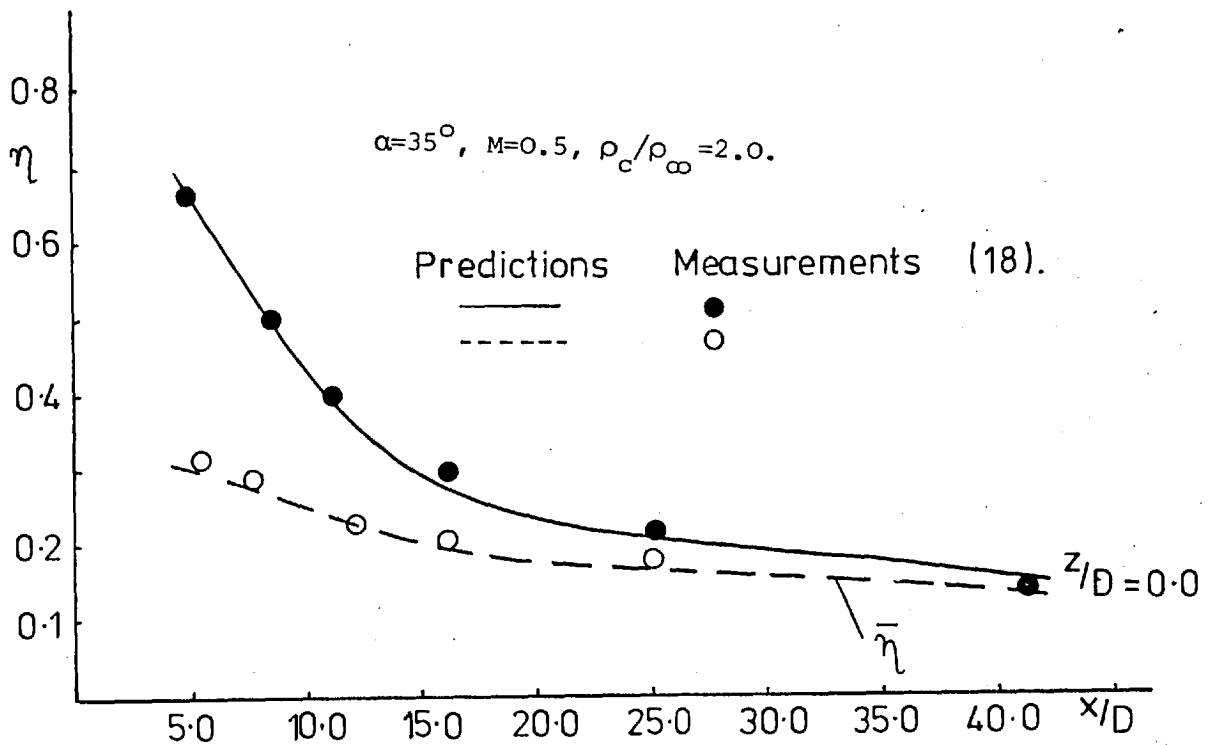


Fig. 76a : Predicted wall film cooling effectiveness comparison with measurements ref. (18),

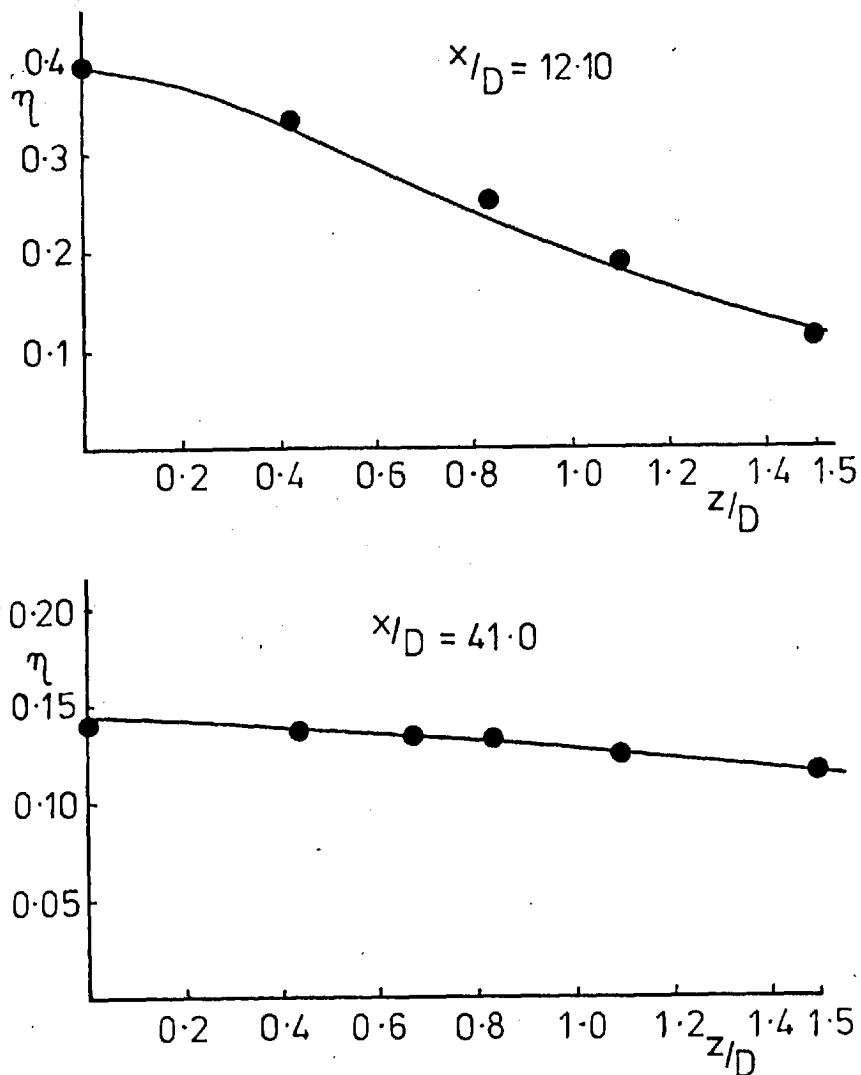


Fig. 76b : Predicted lateral variation of wall effectiveness, comparison with measurements ref. (18), $\alpha=35^\circ$

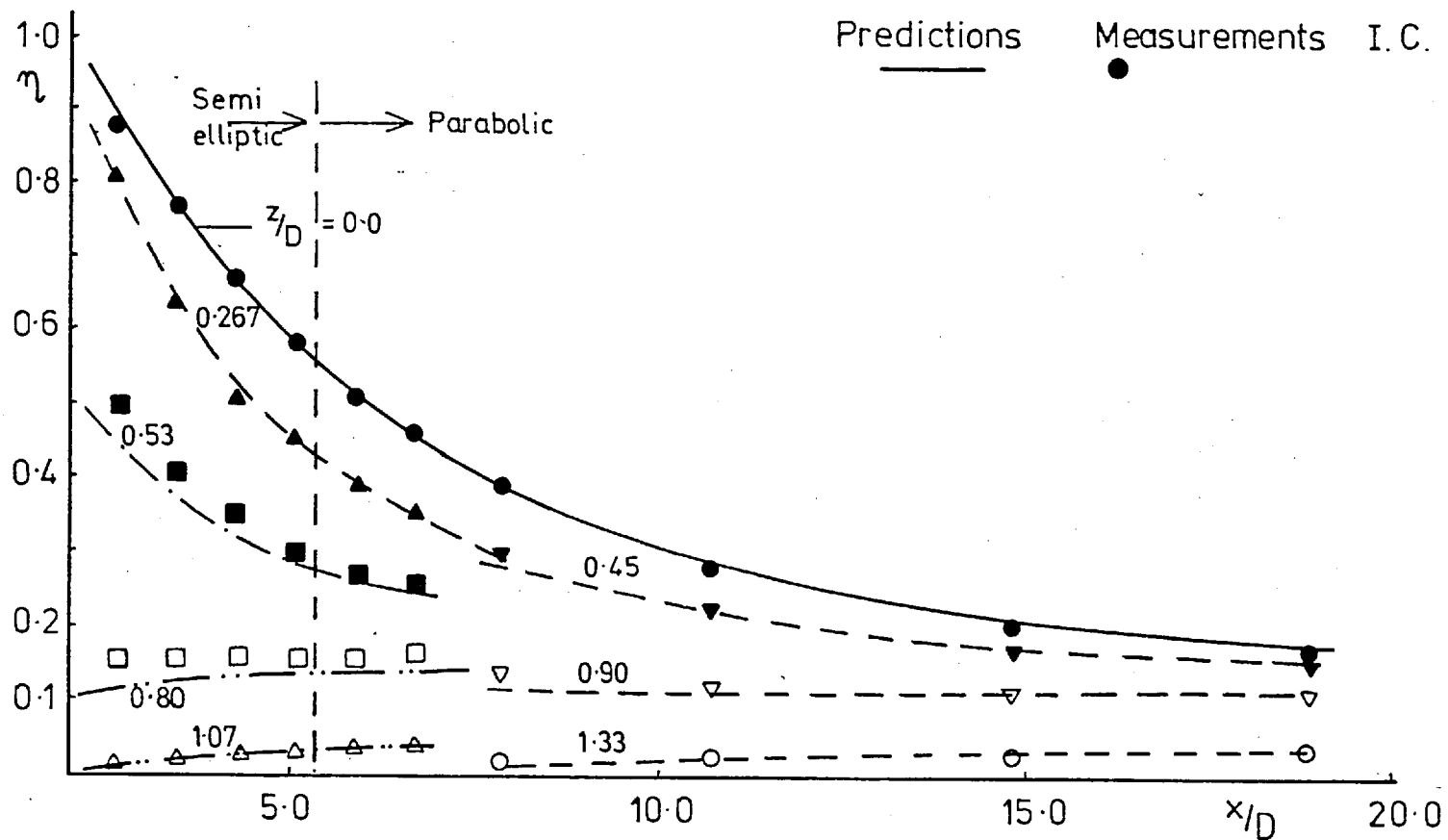


Fig. 77 : Predicted wall film cooling effectiveness,
comparison with measurements IC, $\alpha=30^\circ$,
 $M=0.25$.

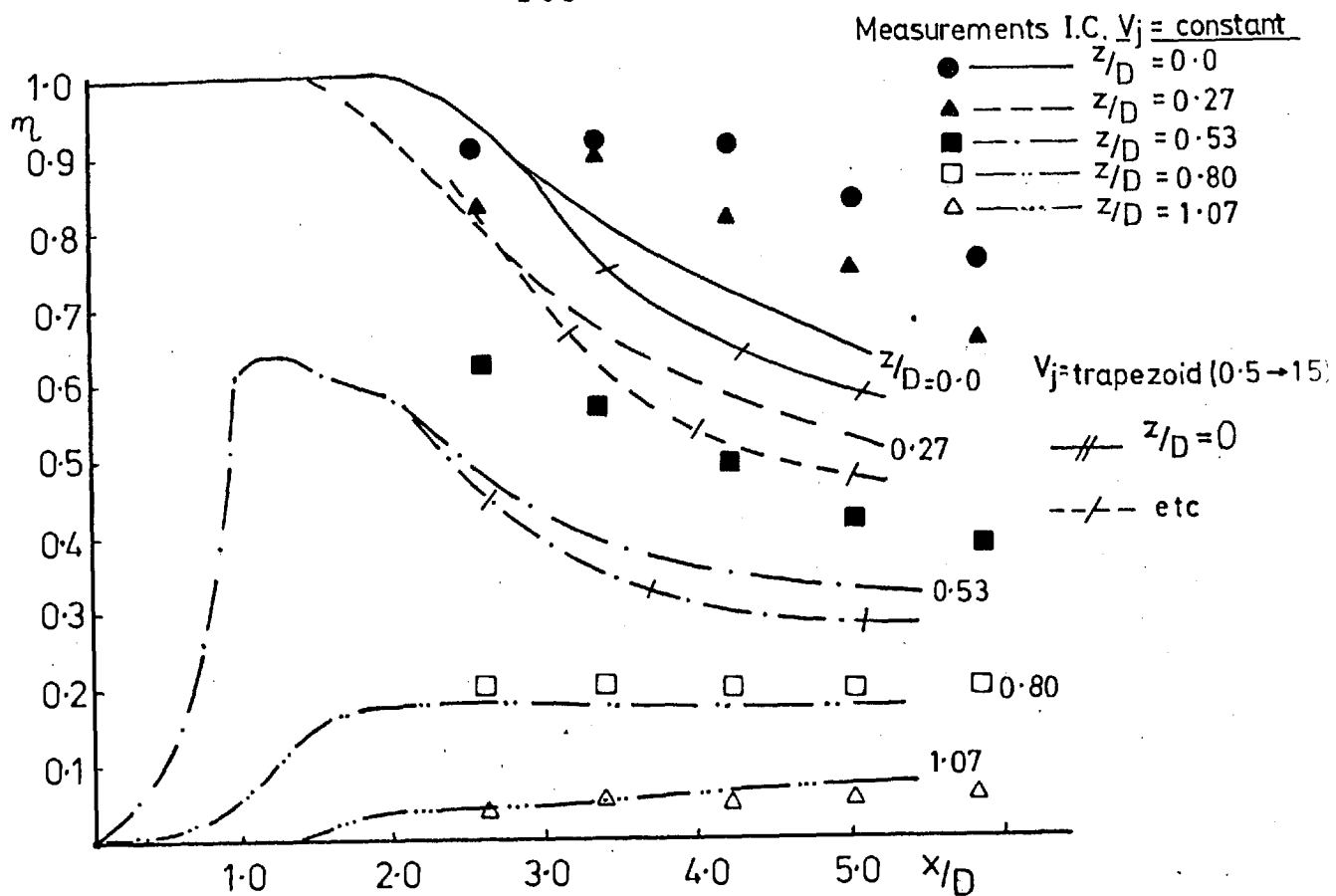


Fig. 78a : Predicted wall film cooling effectiveness,
comparison with measurements IC, $\alpha = 30^\circ$,
 $M = 0.50$.

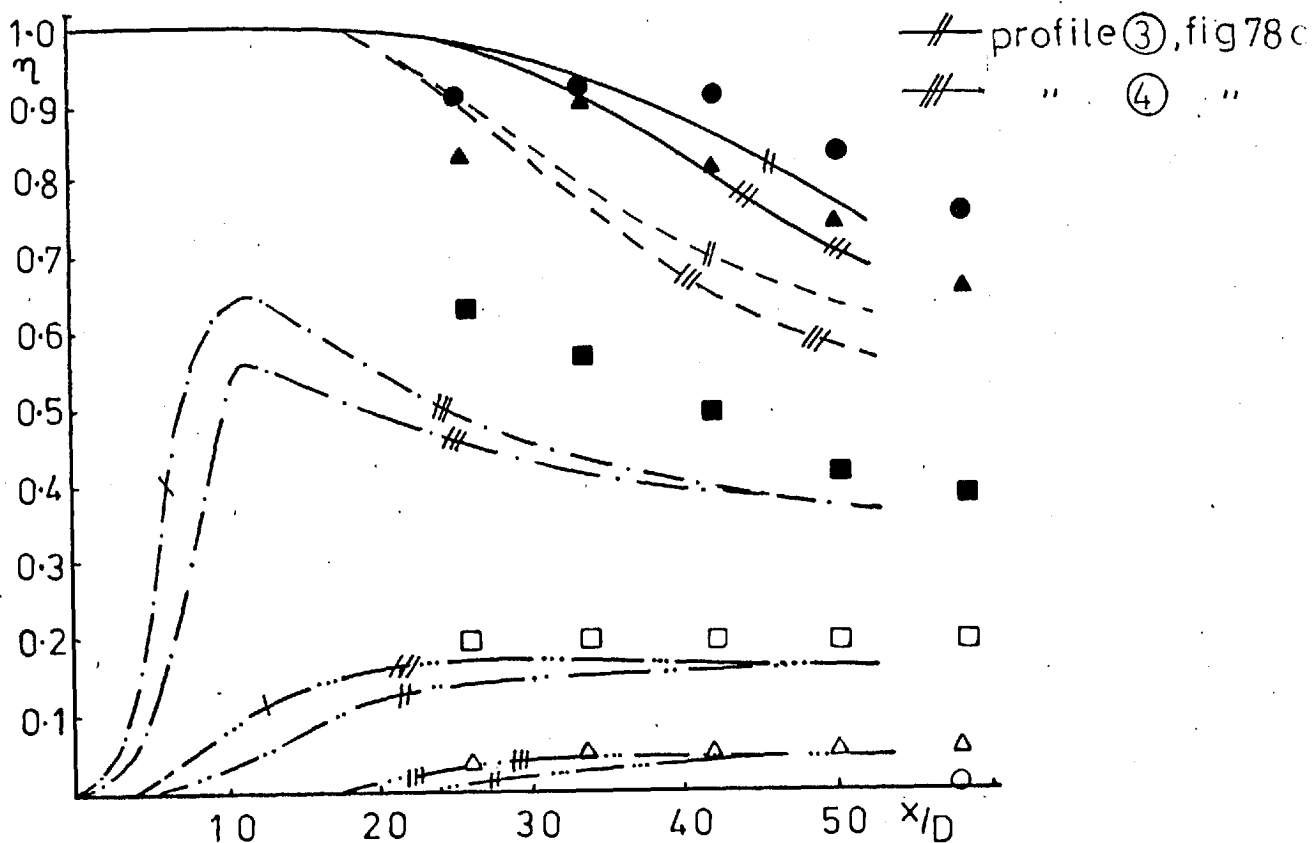


Fig. 78b : Predicted wall film cooling effectiveness,
effect of the jet velocity profile.

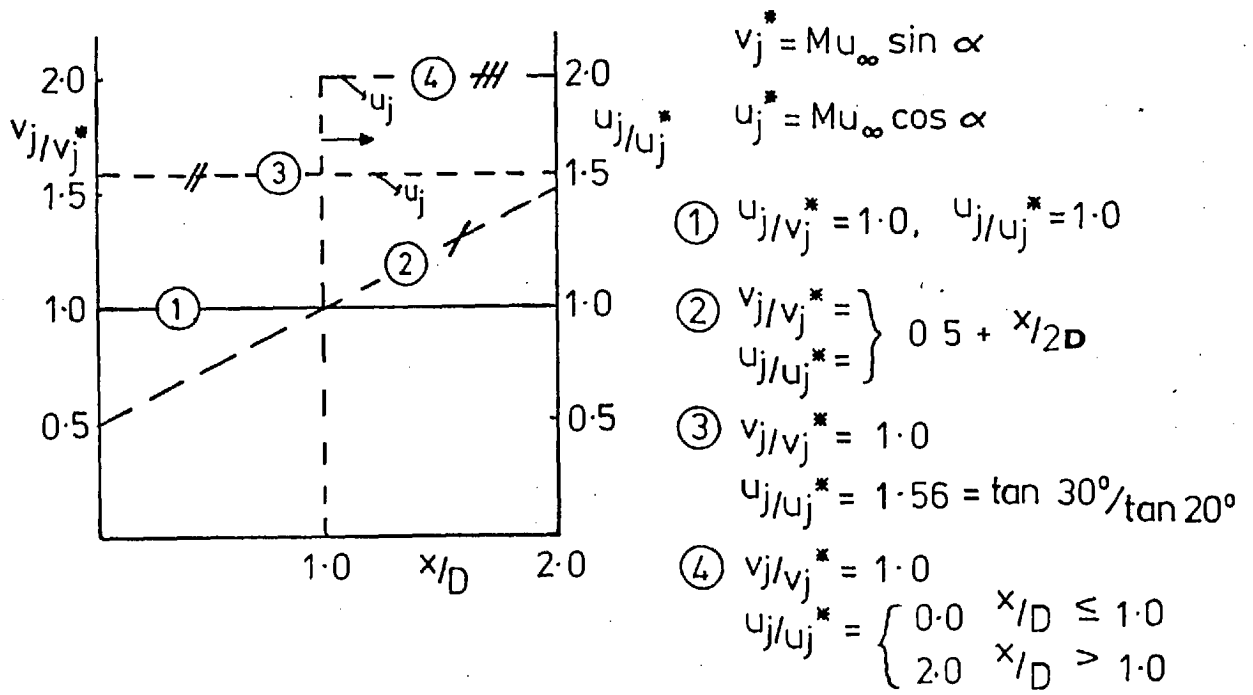


Fig. 78c : Secondary velocity profiles employed for studying its effect on η .

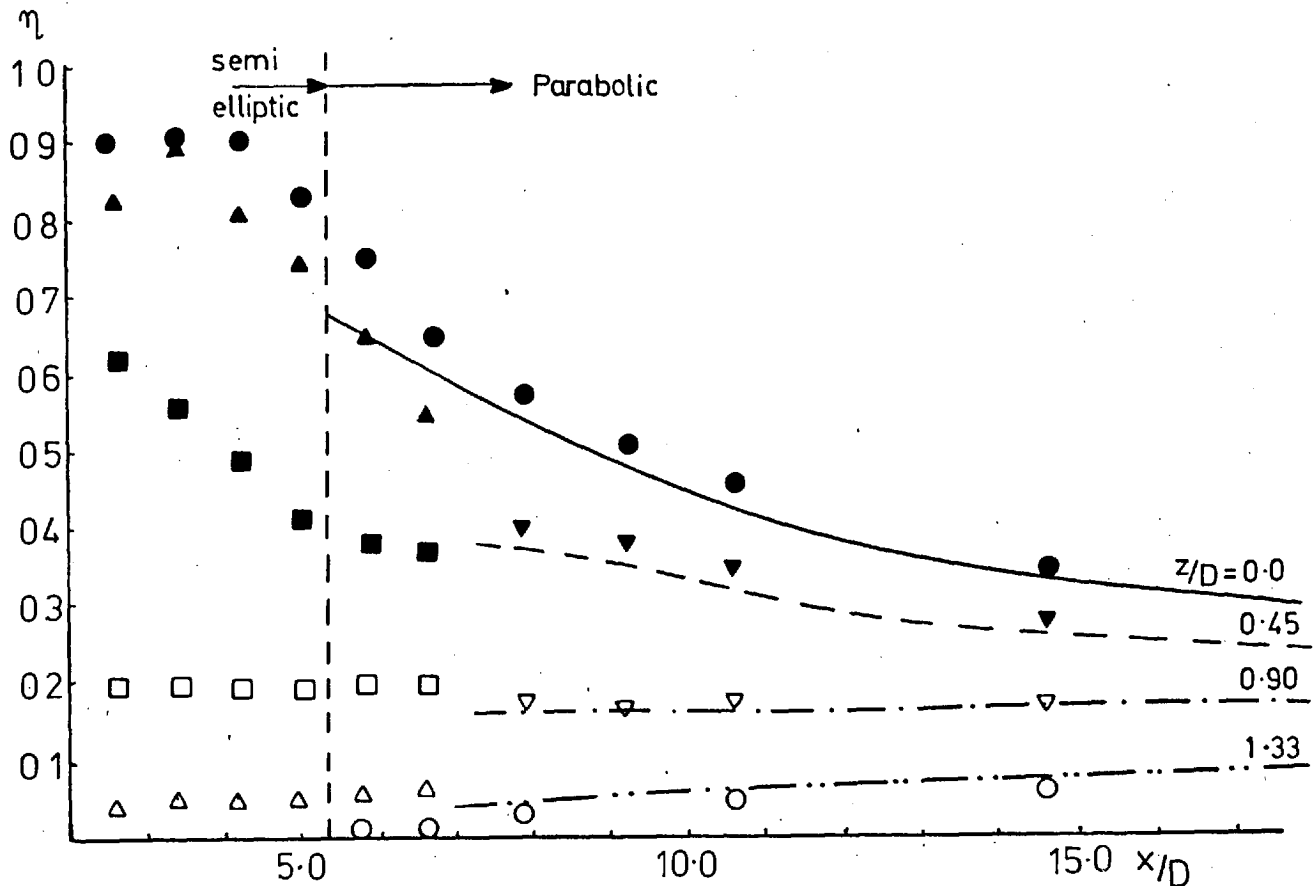


Fig. 78d : Predicted wall film cooling effectiveness, comparison with measurements IC, $\alpha=30^\circ$, $M=0.50$.

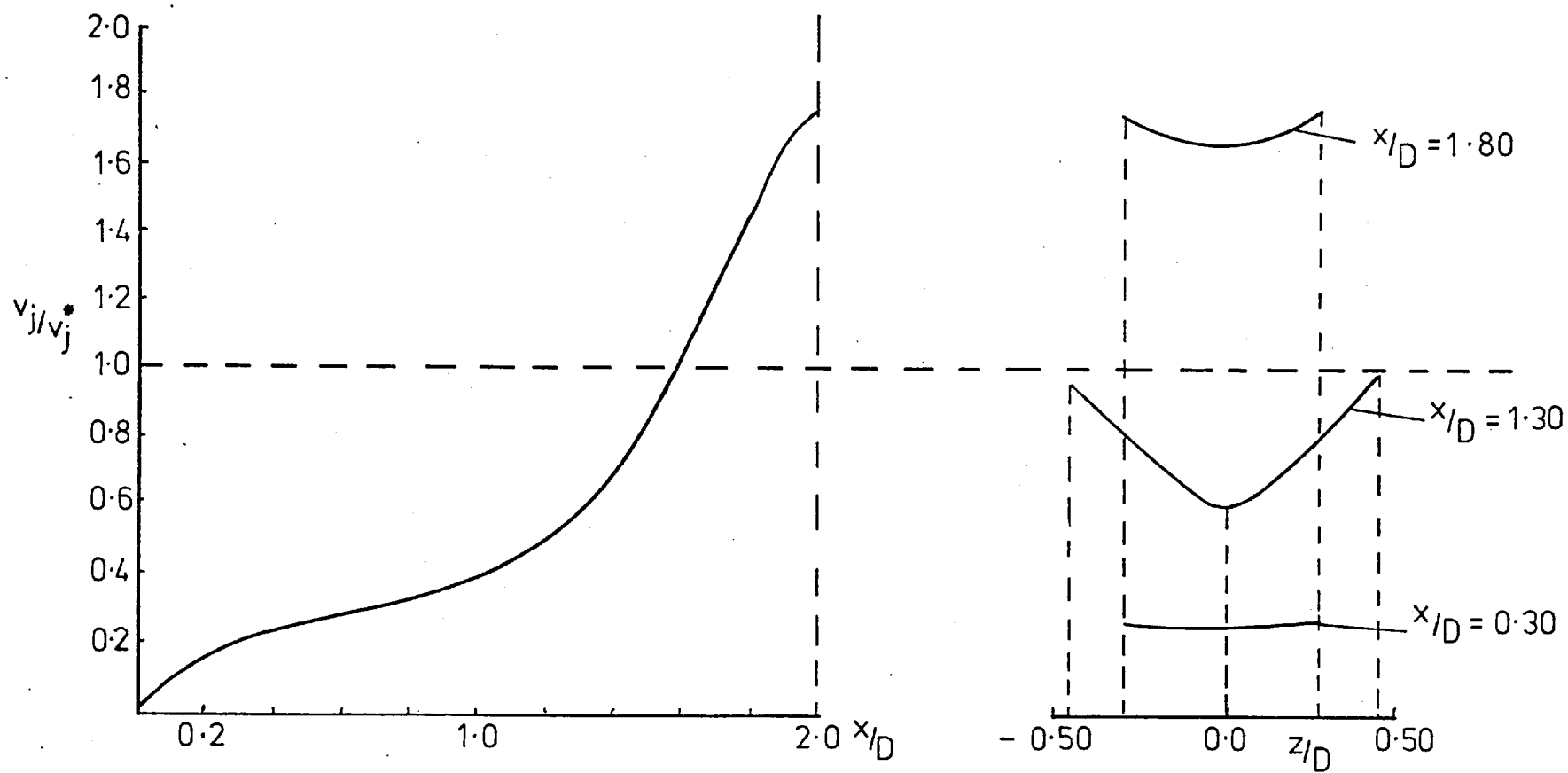


Fig. 79 : Predicted injection velocity profile on
assumption of P_{tot} = constant over the hole.

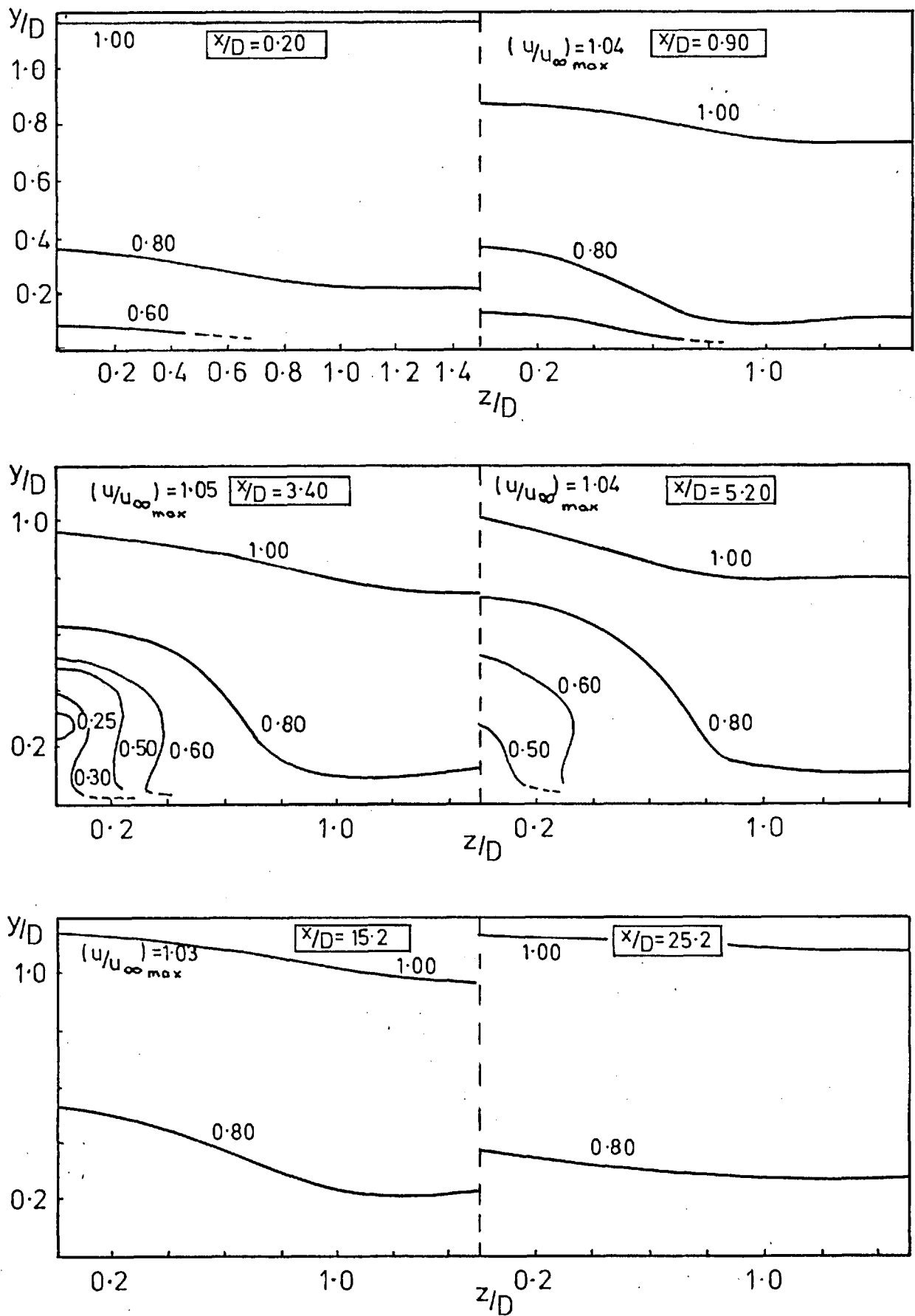


Fig. 80 : Predicted streamwise velocity on cross-stream planes ref. (29), $\alpha=35^\circ$, $M=0.50$.

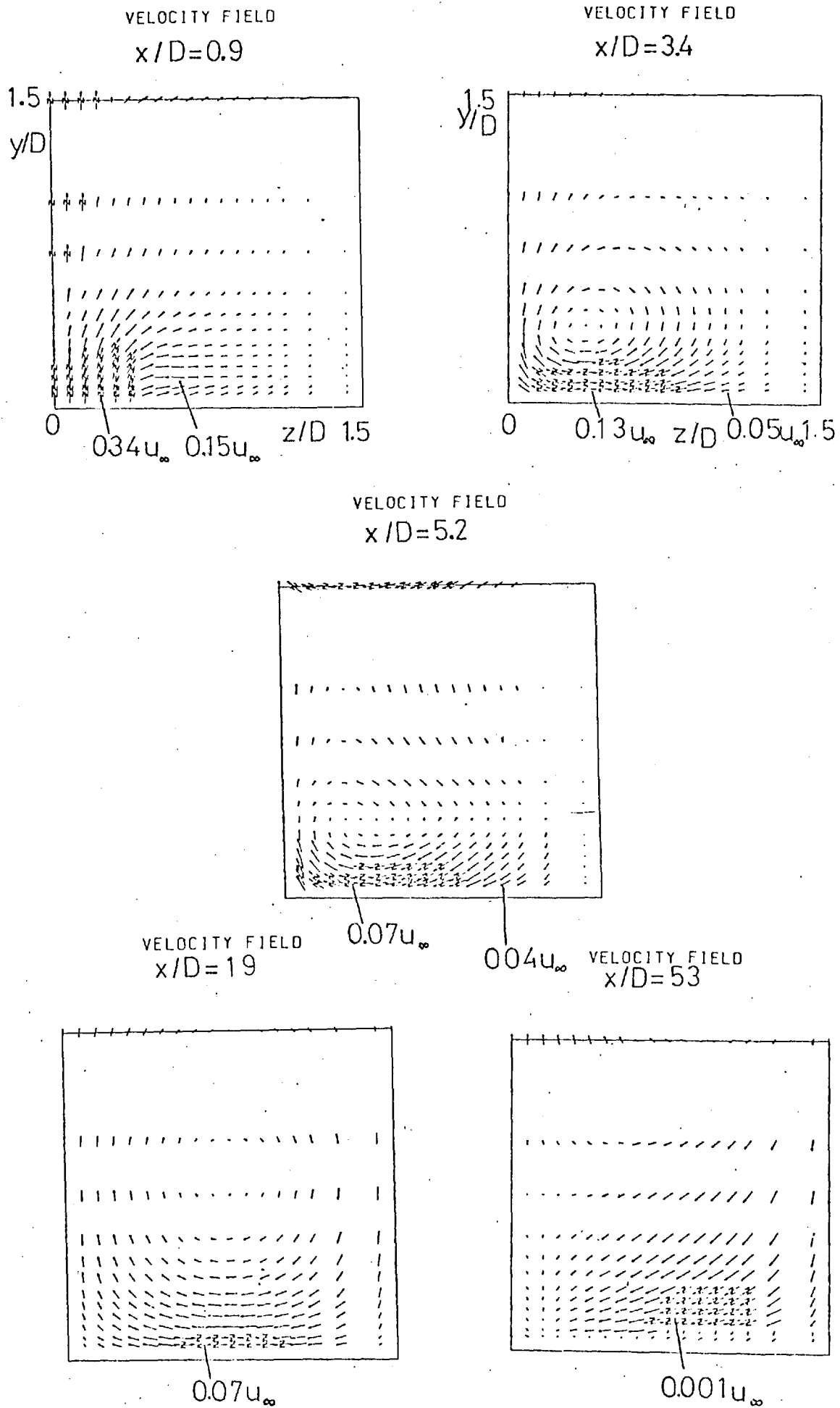


Fig. 81 : Predicted flow pattern on cross-stream planes, ref. (29), $\alpha = 35^\circ$, $M = 0.50$.

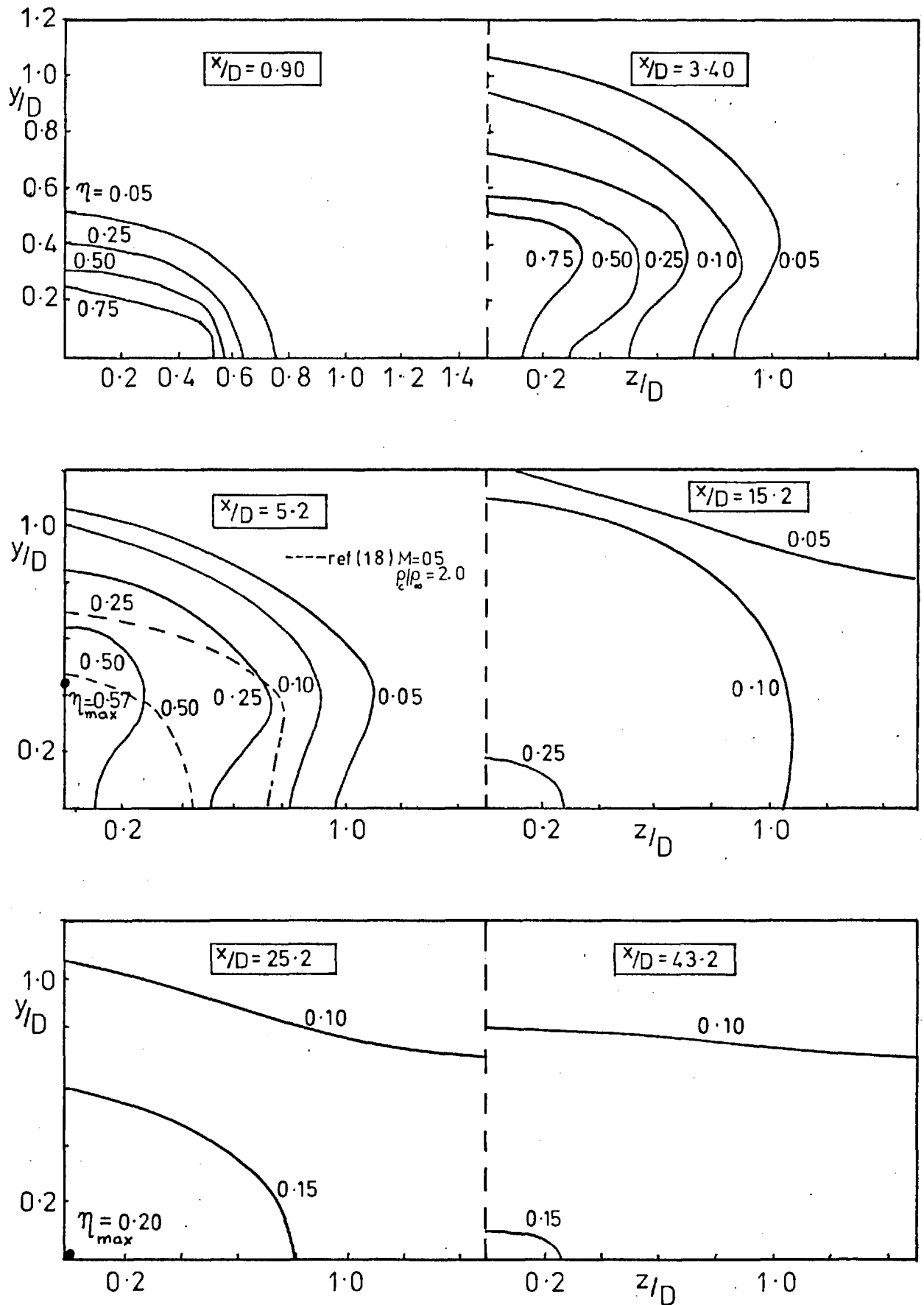


Fig. 82 : Predicted effectiveness on cross-stream planes, ref. (29), $\alpha=35^\circ$, $M=0.50$.

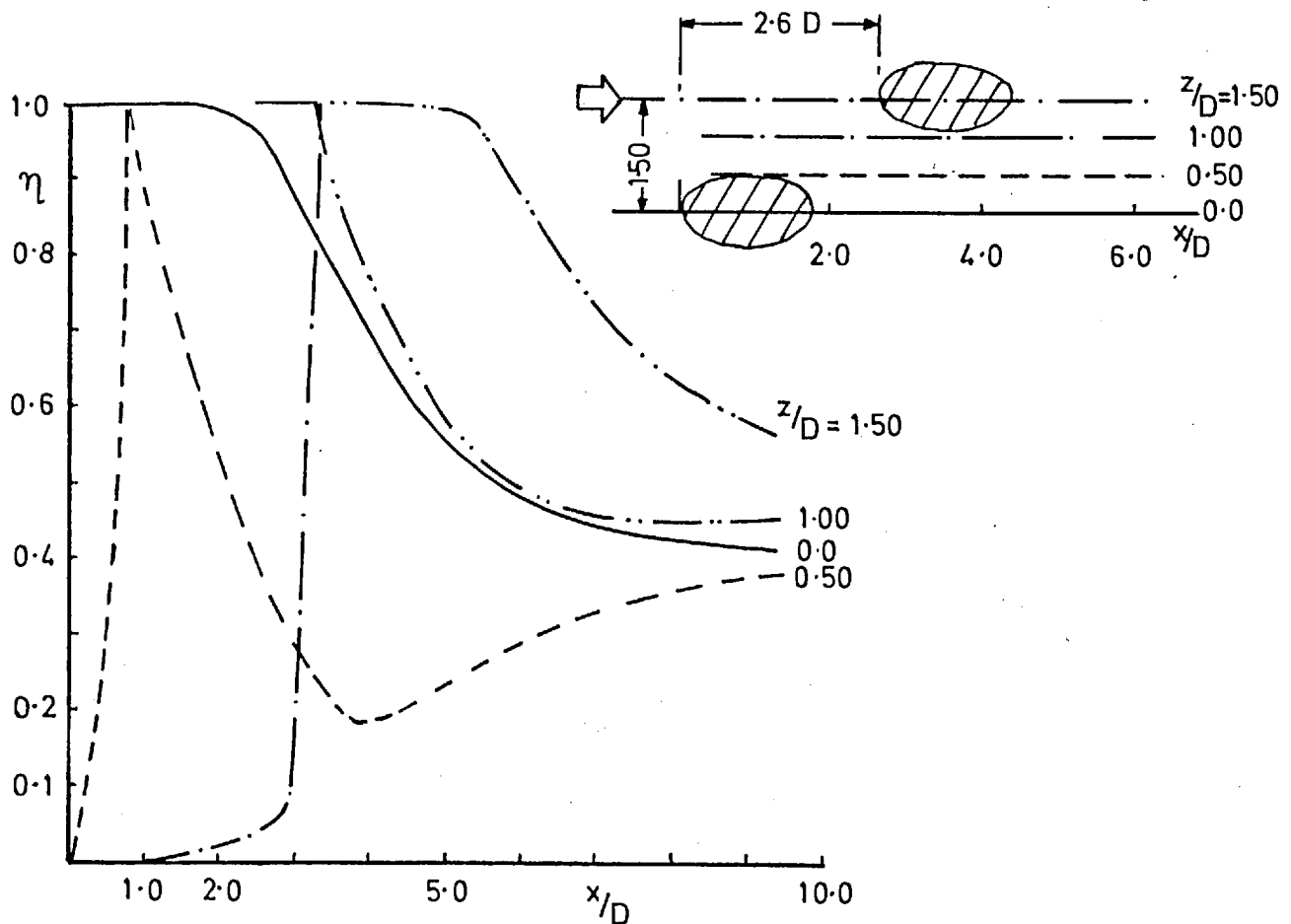


Fig. 83a : Predicted wall film cooling effectiveness,

ref. (25), $\alpha=35^\circ$, $M=0.334$.

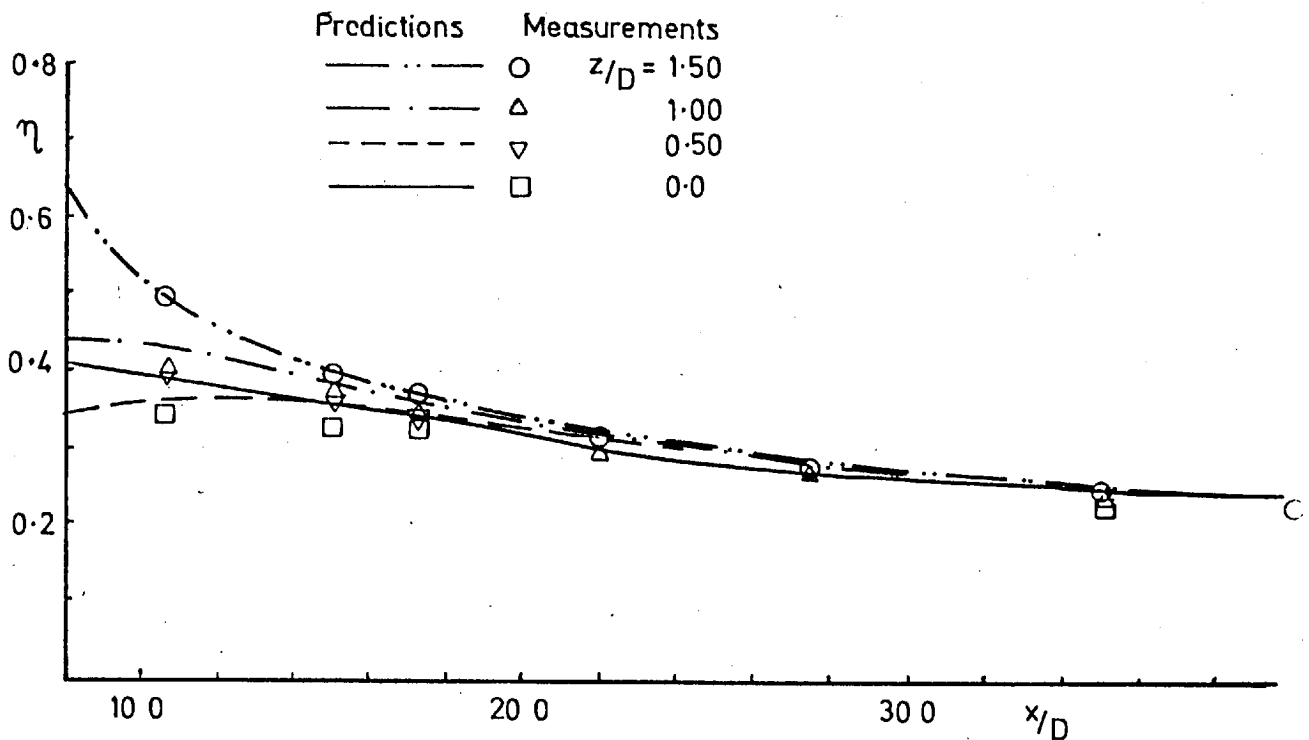


Fig. 83b : Predicted wall film cooling effectiveness,

comparison with measurements, ref. (25)

$\alpha=35^\circ$, $M=0.334$.

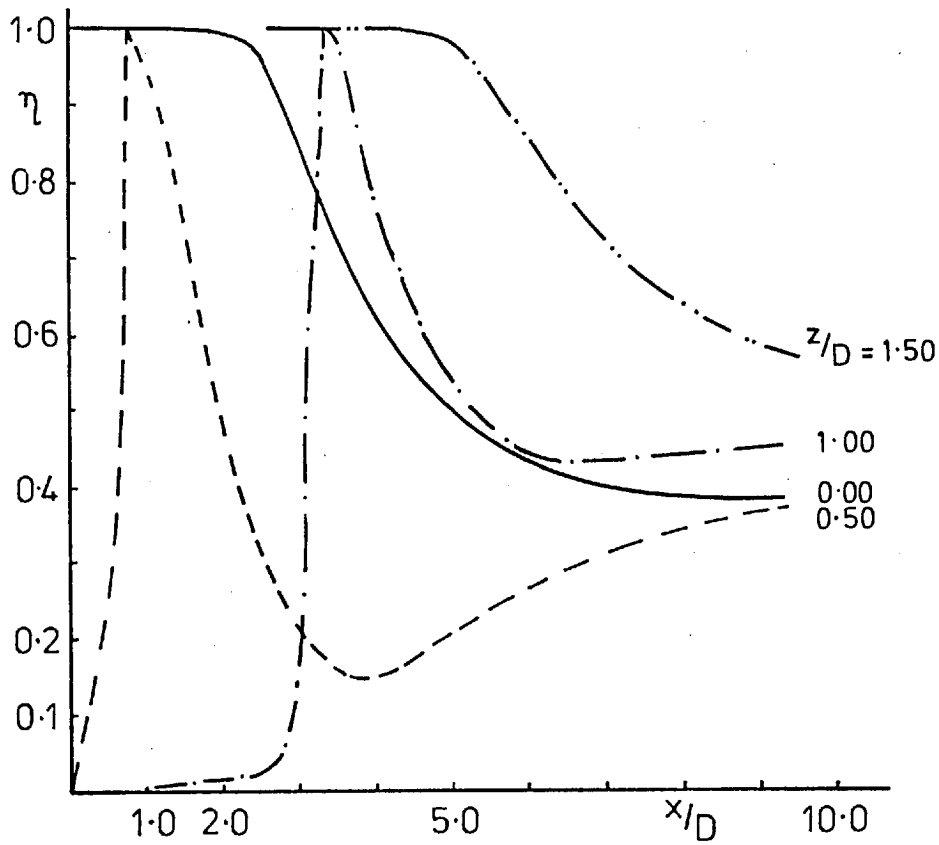


Fig. 84a : Predicted wall film cooling effectiveness

ref. (25), $\alpha=35^\circ$, $M=0.47$.

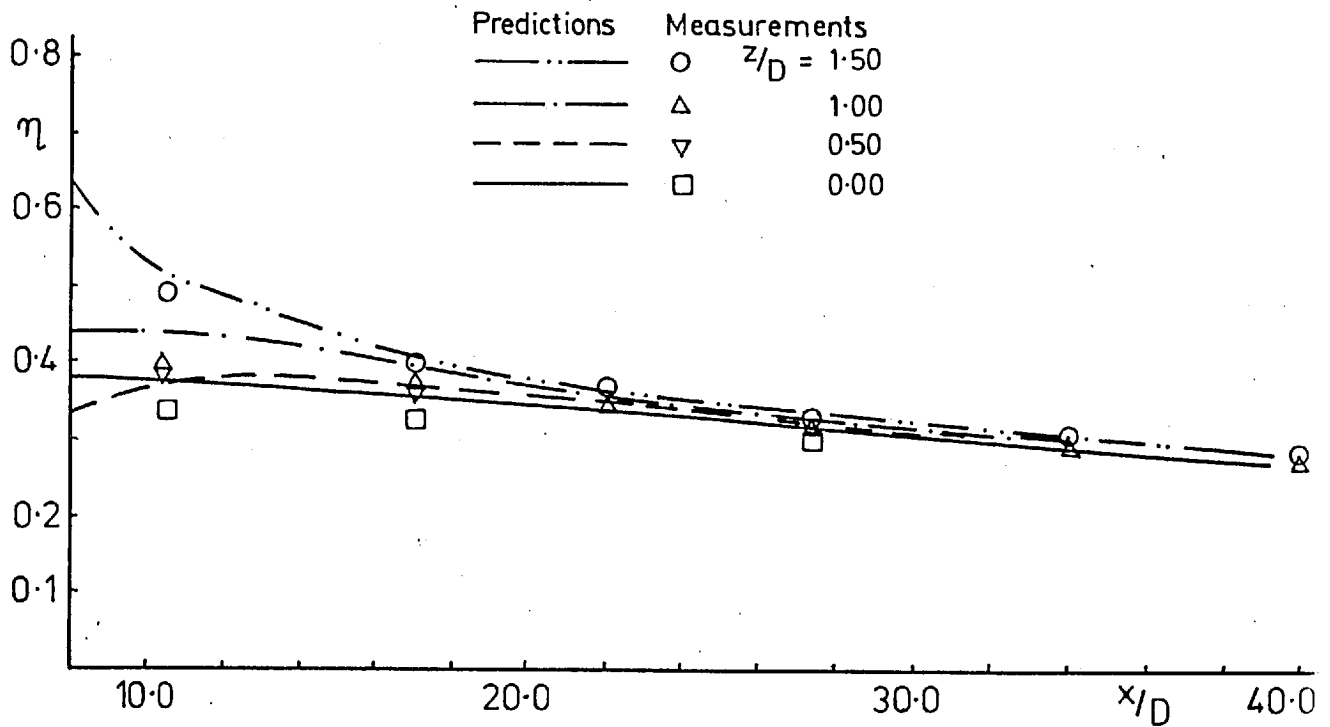


Fig. 84b : Predicted wall film cooling effectiveness,

comparison with measurements ref. (25)

$\alpha=35^\circ$, $M=0.47$.

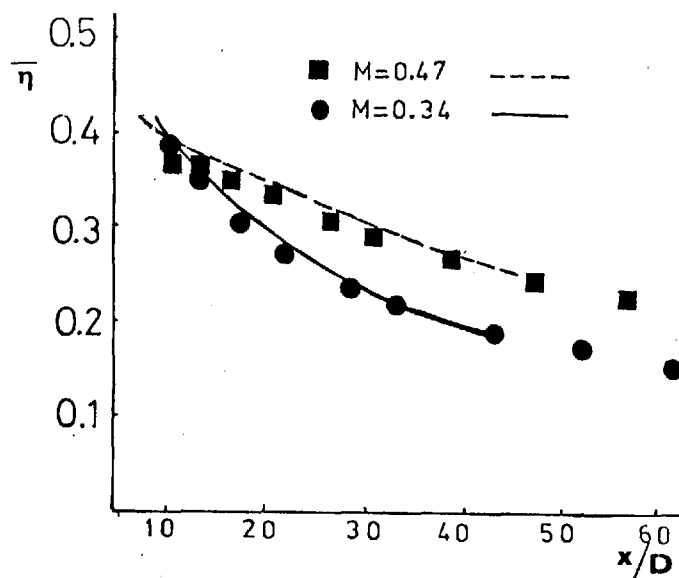


Fig. 85.: Predicted averaged wall effectiveness,
comparison with measurements, ref. (25),
 $\alpha=30^\circ$.

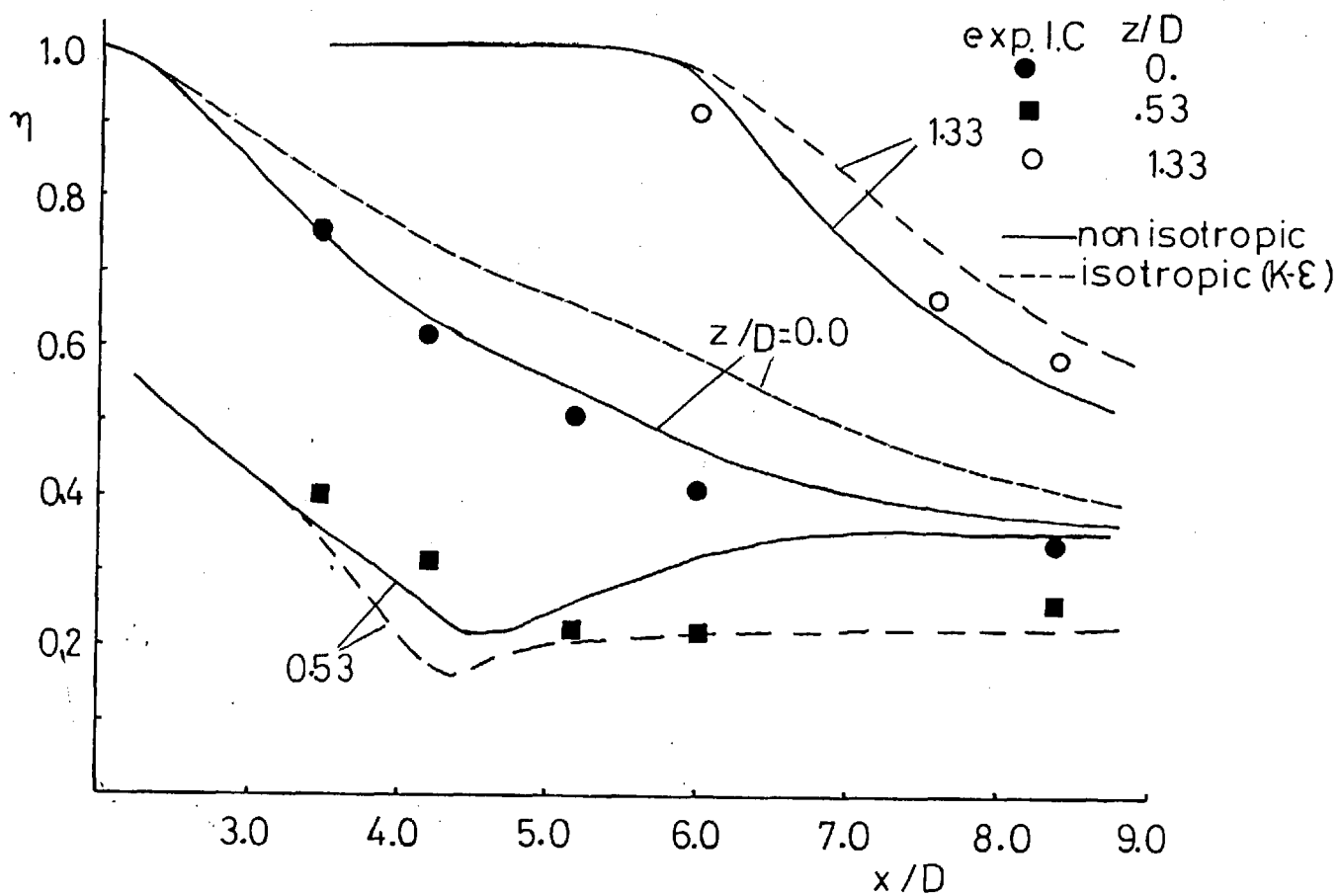


Fig. 86a : Predicted wall effectiveness, comparison
with measurements IC, $\alpha=30^\circ$, $M=0.25$.

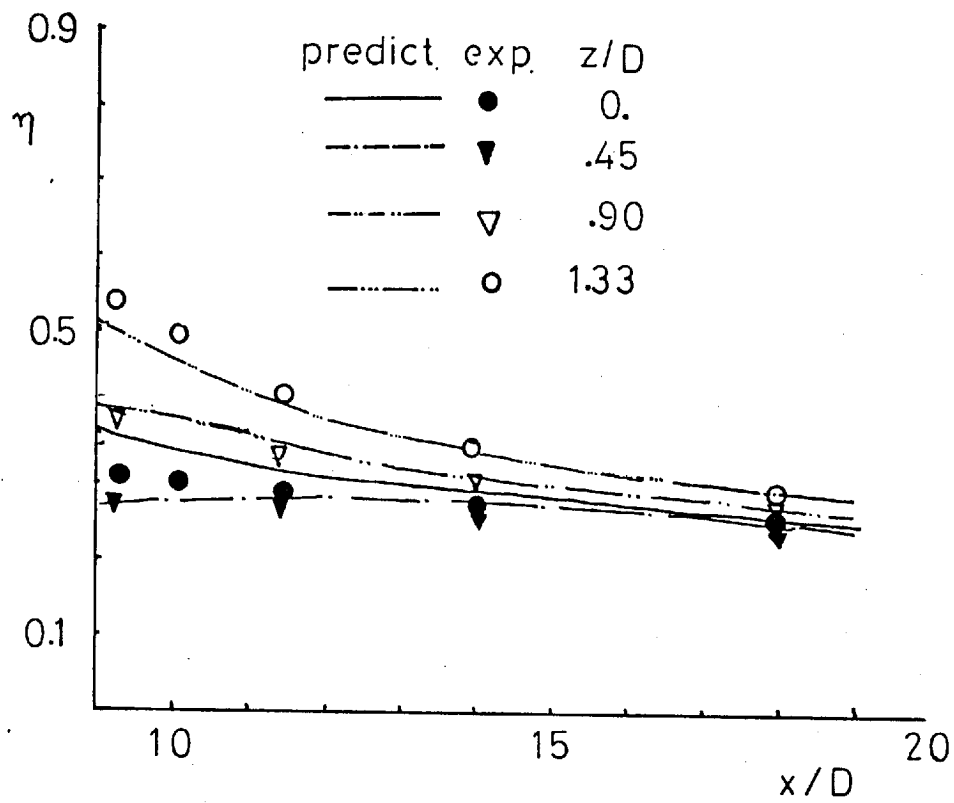


Fig. 86b : Predicted wall effectiveness, comparison
with experiments IC, $\alpha=30^\circ$, $M=0.25$.

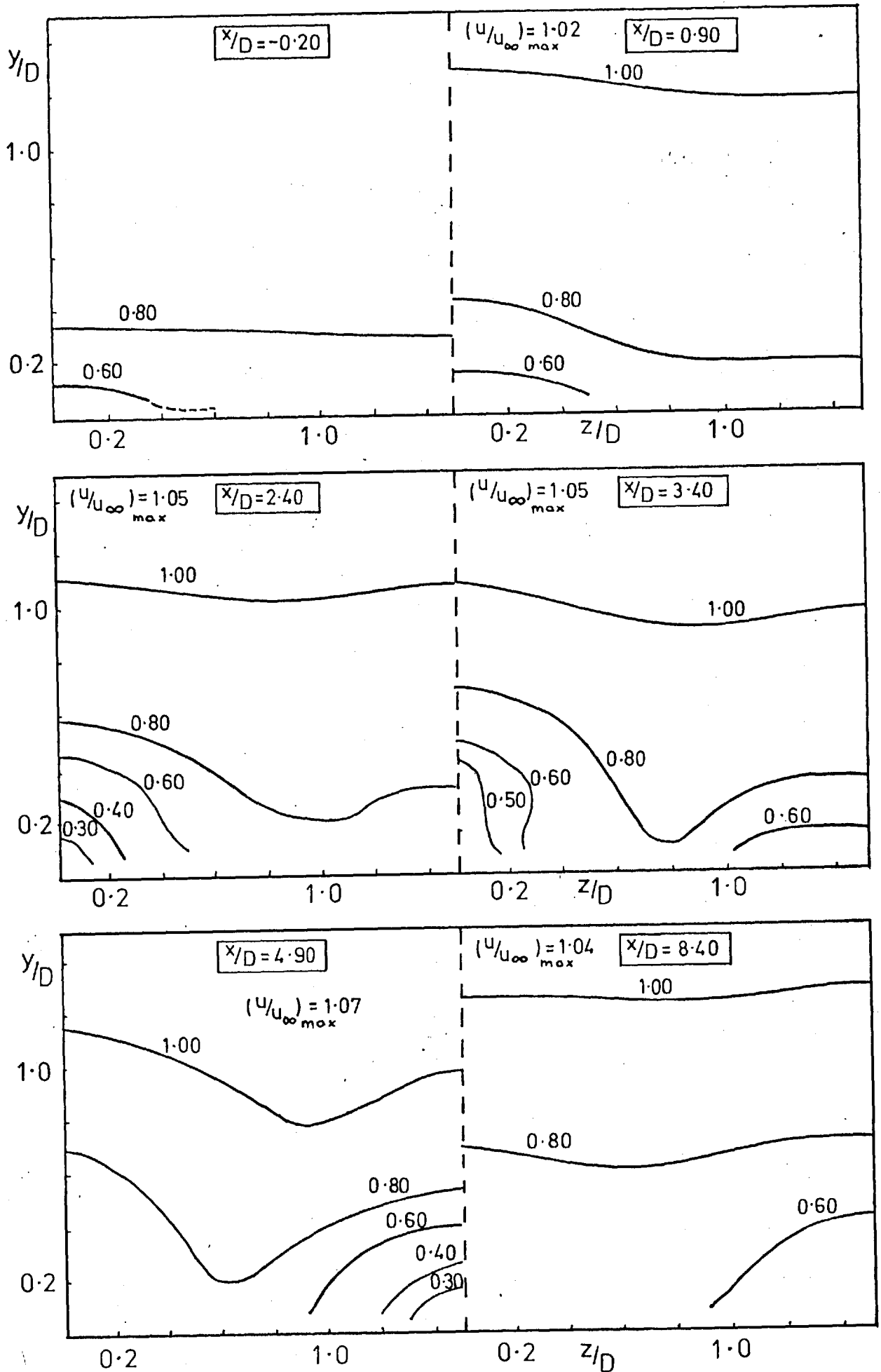


Fig. 87 : Predicted streamwise velocity on cross-

stream planes ref. (25), $\alpha=35^\circ$, $M=0.47$.

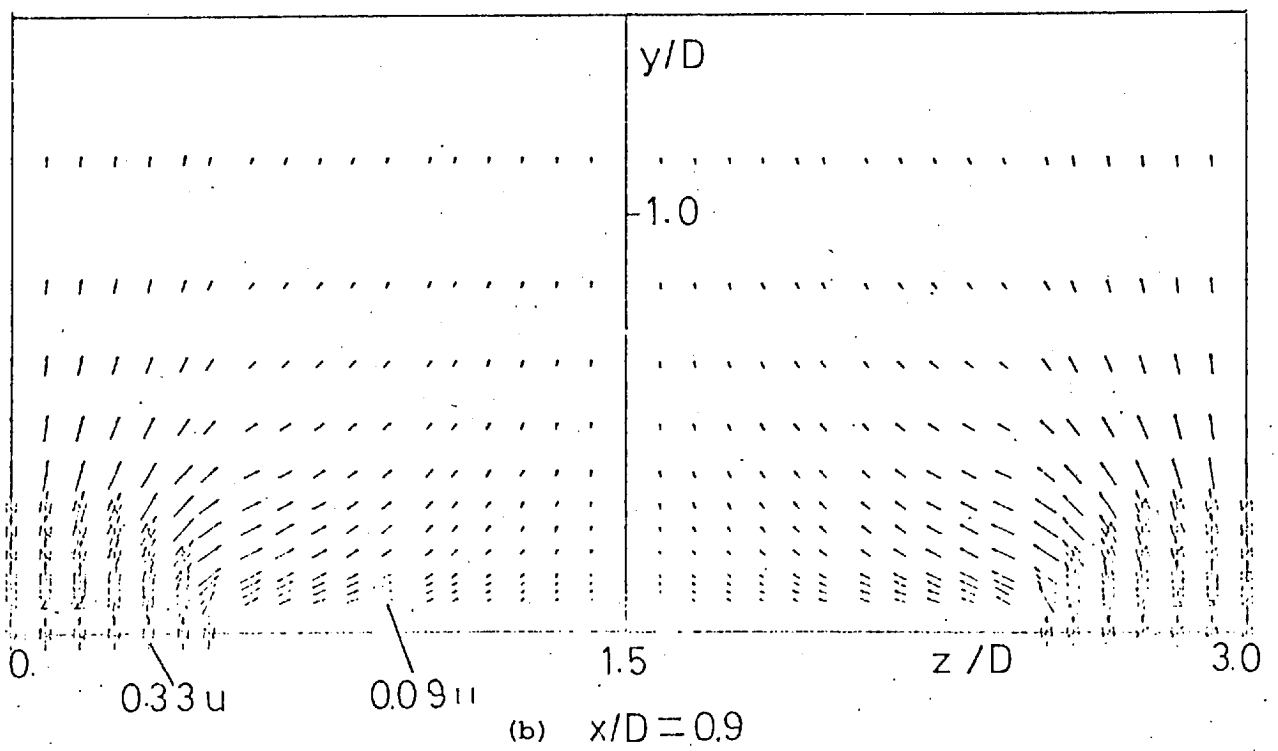
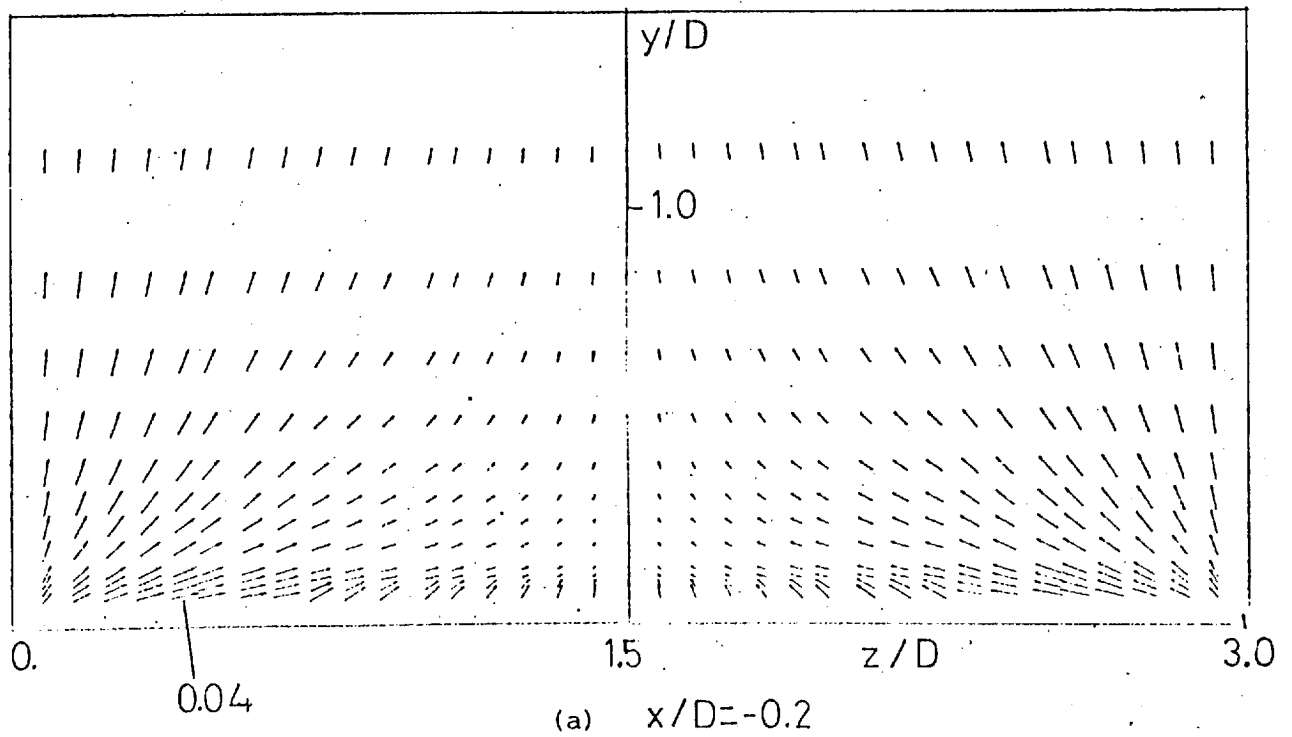


Fig. 88 : Predicted flow field on cross-stream
planes, ref. (25), $\alpha=35^\circ$, $M=0.47$.

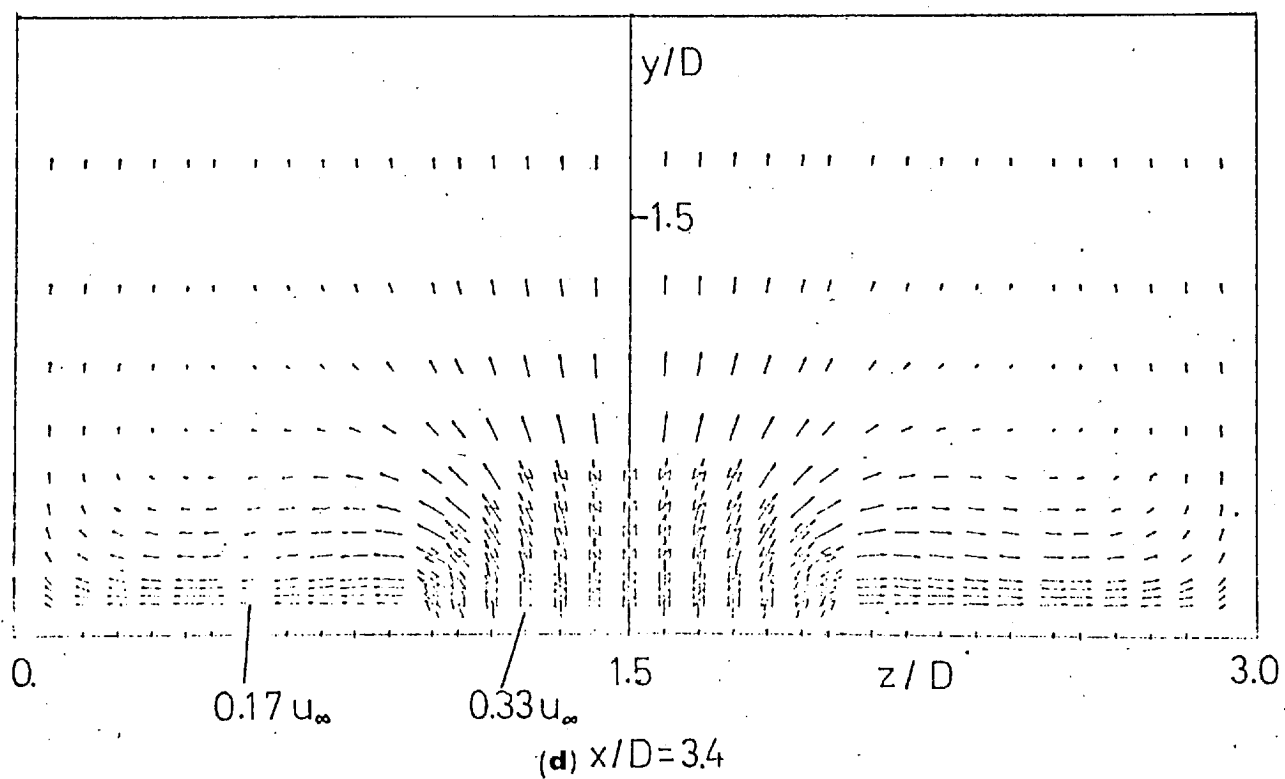
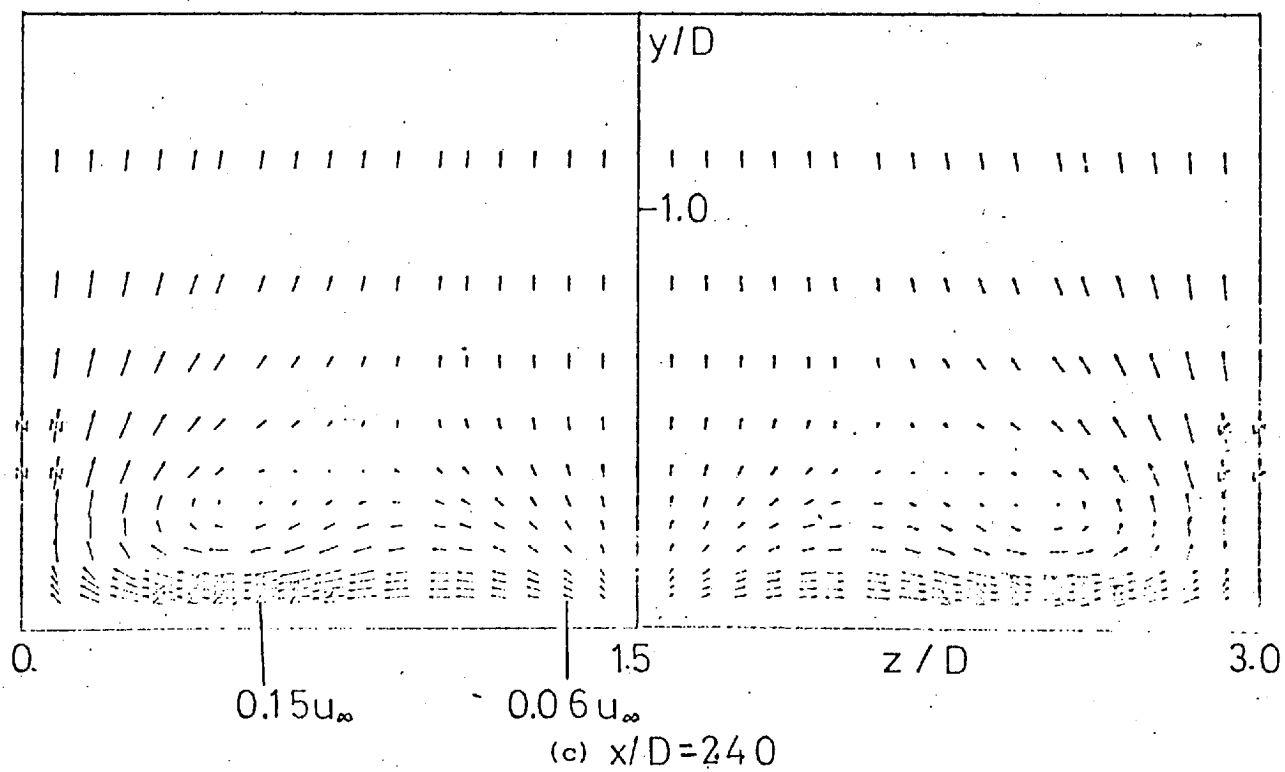


Fig. 88 : Predicted flow field on cross-stream planes

ref. (25), $\alpha = 35^\circ$, $M = 0.47$.

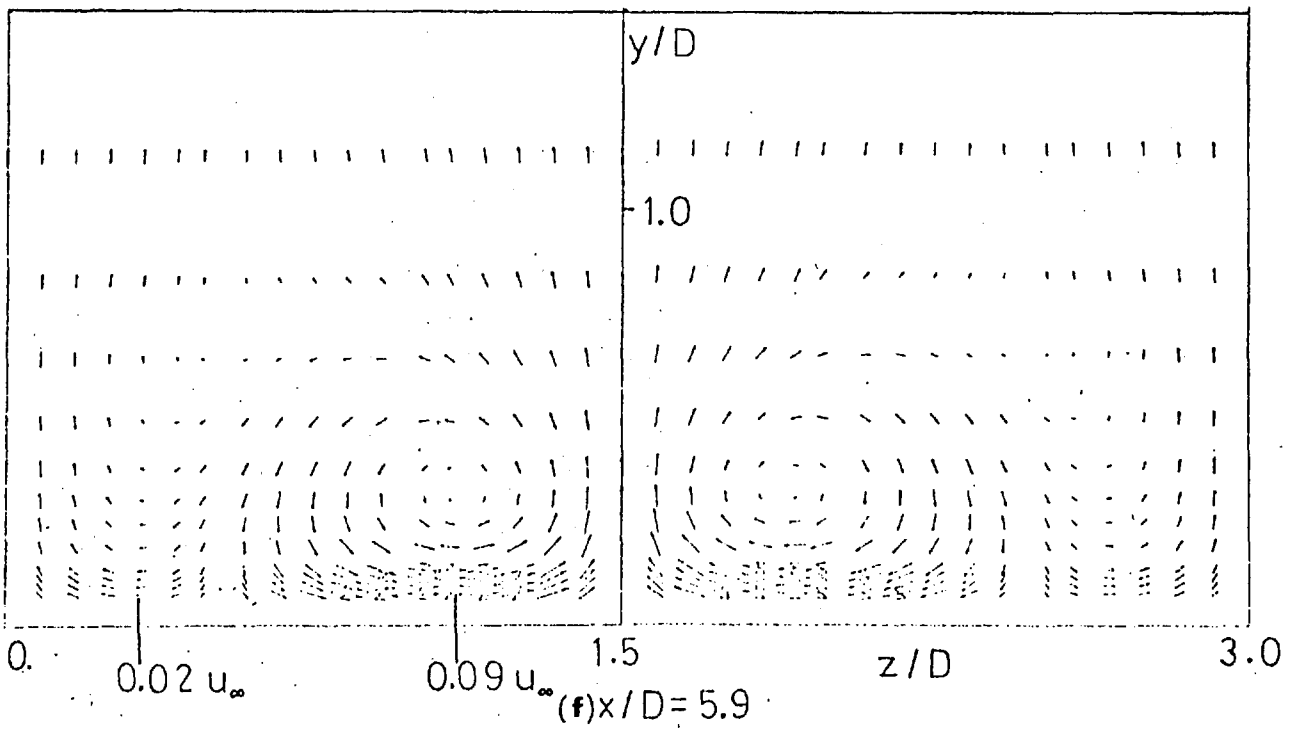
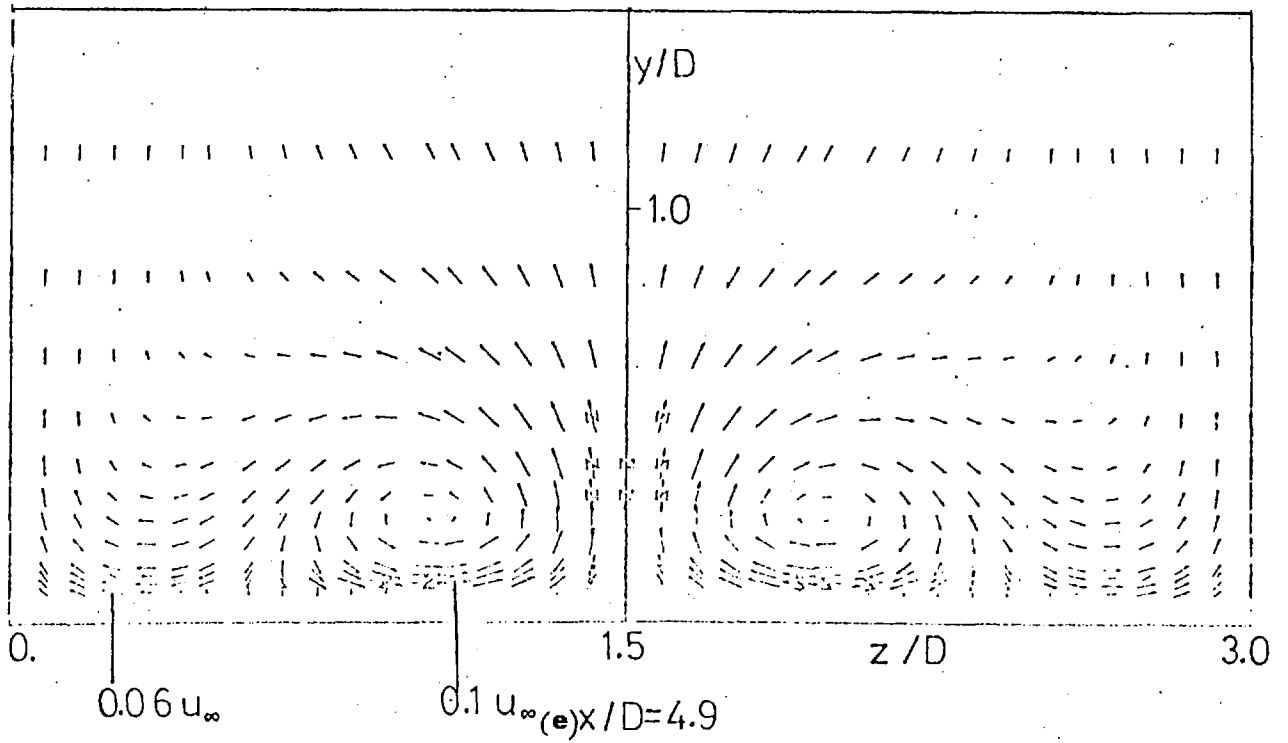


Fig. 88 : Predicted flow field on cross-stream planes,
ref. (25); $\alpha=35^\circ$, $M=0.47$.

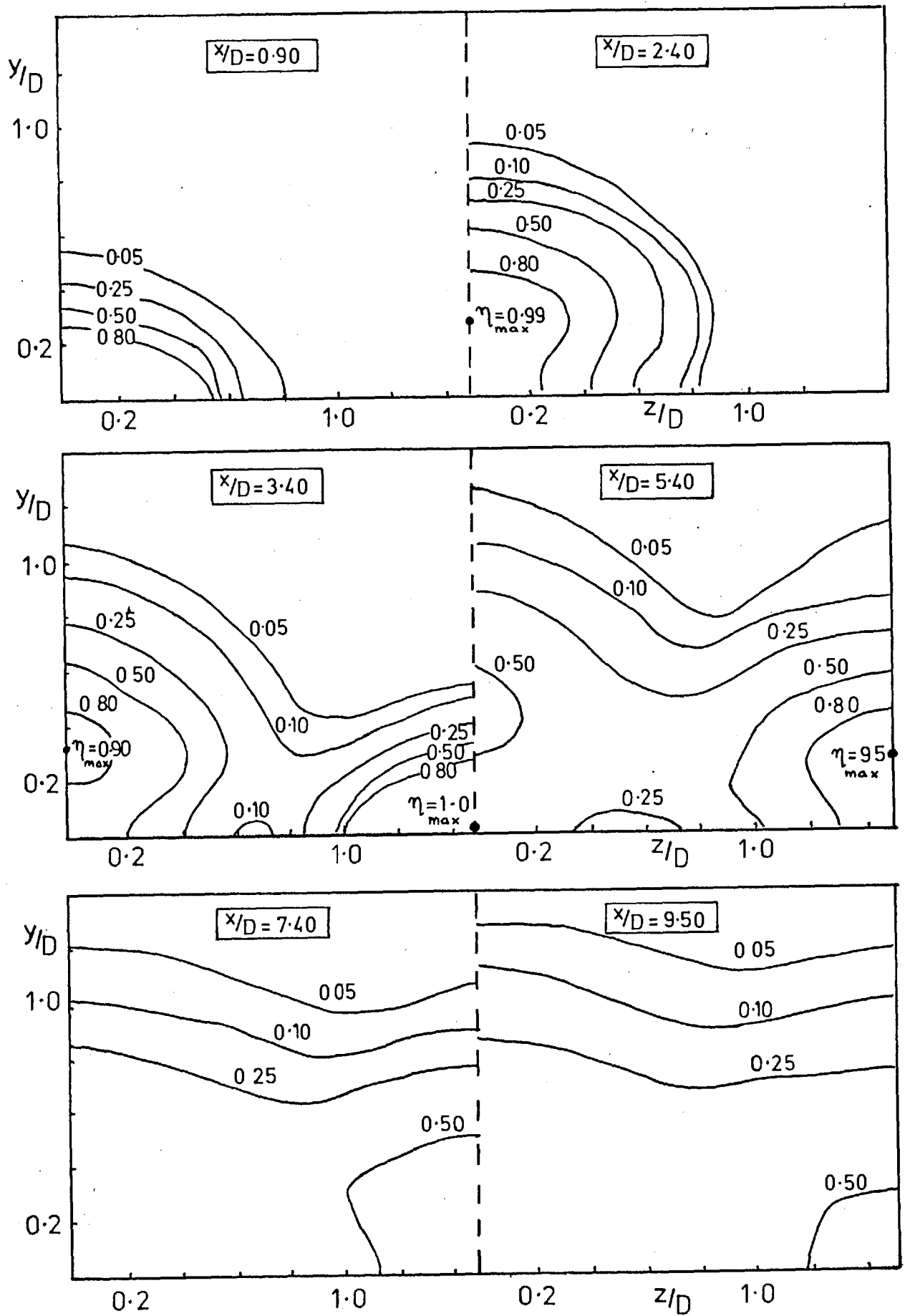


Fig. 89a : Predicted effectiveness contours on cross-stream planes, ref. (25), $\alpha = 35^\circ$, $M = 0.47$.

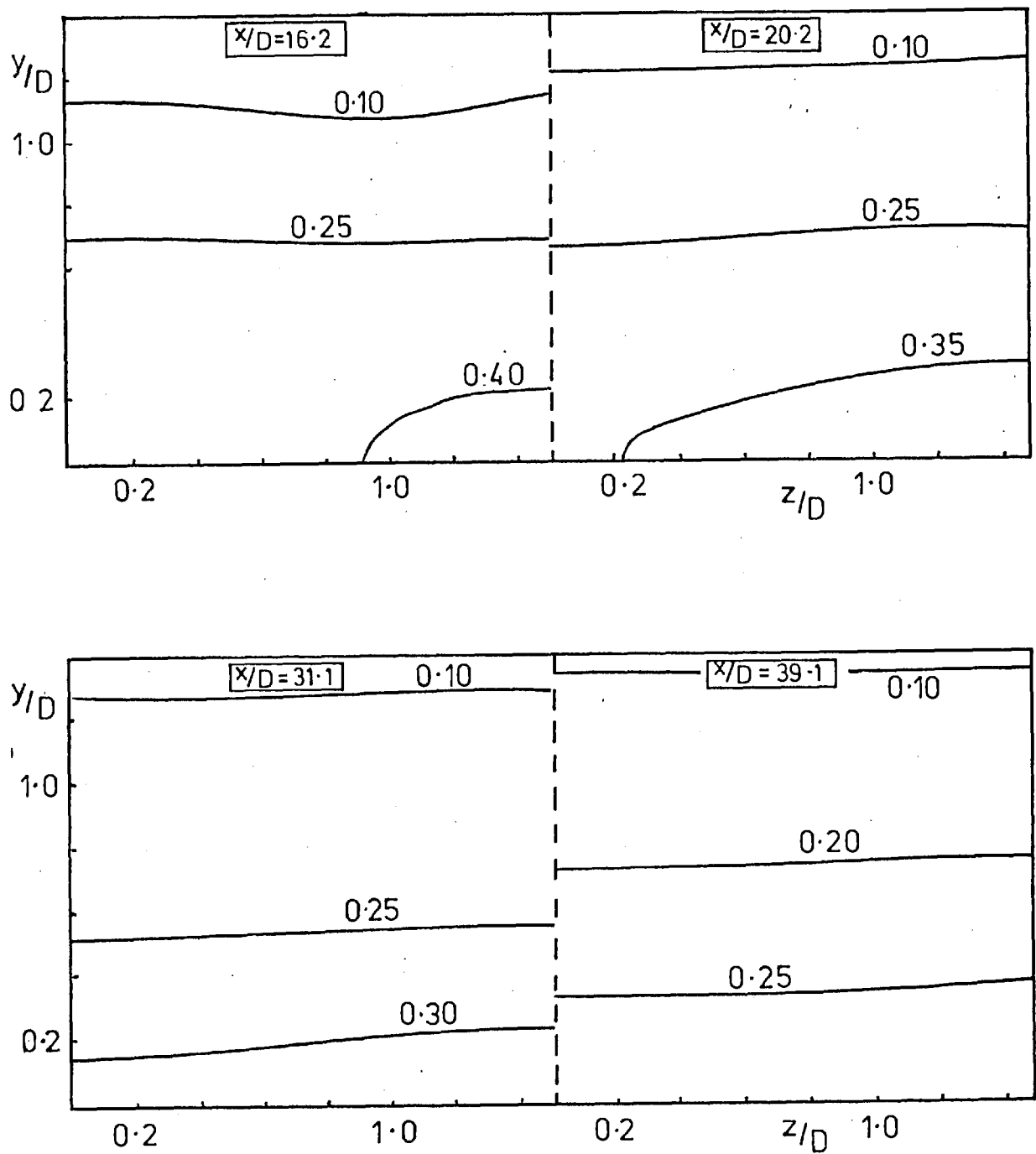


Fig. 89b : Predicted effectiveness contours on cross-stream planes, ref. (25), $\alpha = 35^\circ$, $M = 0.47$.