

**TRANSIENT TURBULENT JETS OF
MISCIBLE AND IMMISCIBLE FLUIDS**

BY

S T XI

1986

THESIS SUBMITTED FOR THE DEGREE OF
DOCTOR OF PHILOSOPHY
IN THE FACULTY OF ENGINEERING
UNIVERSITY OF LONDON
AND
FOR THE DIPLOMA OF MEMBERSHIP OF
IMPERIAL COLLEGE

Computational Fluid Dynamics Unit
Room 440, MED Building
IMPERIAL COLLEGE OF SCIENCE AND TECHNOLOGY
Exhibition Road
LONDON SW7 2BX
U.K.

TO MY WIFE HUI-JU

ABSTRACT

The investigation of transient turbulent jets of miscible and immiscible fluids is the main subject of this thesis.

The quasi-two-phase nature of miscible turbulent jets and the true two-phase nature of immiscible ones are analogous. As a consequence of the analogy, both jets are simulated by combining the constant-effective-viscosity model with some assumptions of the two-fluid model. The simulated jets include those of miscible and immiscible fluids, with or without buoyancy, and with constant or time-dependent boundary conditions.

An experimental rig is designed and constructed. Similarities between miscible (brine-into-water) and immiscible (kerosene-into-water) transient, turbulent jets are observed: both jets grow in a very similar manner.

Tip-penetration rates of the two transient jets are measured from a visual examination of the successive photographic negatives taken during the experiments.

The droplet-size distribution of the kerosene-into-water jet under steady-state conditions is also measured. A mean droplet diameter is calculated from the droplet-size distribution and this is supplied as a parameter for the

modelling assumptions in the mathematical model.

The computational results are compared with the experimental data of other authors as well as the present study. Good agreement between them is obtained. Recommendations on the modelling assumptions and empirical constants are also made.

PREFACE

I joined the Computational Fluid Dynamics Unit of Imperial College in October, 1982. The work presented in this thesis is a major part of my research activities between then and September, 1985. This work is mainly concerned with the investigation of transient, turbulent jets of miscible and immiscible fluids.

During the first year at the unit, I followed the courses of "Turbulence models", "Mathematical modelling of fluid-mechanics, heat-transfer and chemical-reaction processes", and "Combustion theory applied to engineering" given by the unit and was engaged in the studying and application of the PHOENICS computer code of Spalding (1981) as a preliminary step towards familiarising myself with the turbulence model and numerical methods used in the unit. The flows, which I studied, included a laminar diffusion flame and a heavy gas dispersion in unconfined space (with Z Y Wu).

The main research started with the experimental investigation of the droplet size in turbulent two-phase flow. The droplet size of a kerosene-into-water steady, turbulent jet was measured. The attention was then turned to studying the behaviour of the miscible and immiscible transient, turbulent jets by using experimental and computational methods. The same rig was used to record a

brine-into-water and a kerosene-into-water transient, turbulent jets while the modelling assumptions were incorporated into the PHOENICS code to simulate the flows considered. The piece of work presented in this thesis was completed by the end of September, 1985.

I would like to take this opportunity to express my gratitude to all those who have contributed to my studies at Imperial College. First of all, I am most grateful to my supervisor, Professor D B Spalding for the skilful guidance and supervision throughout the course of my study. It is indeed a privilege to have worked with him. I would also like to thank Dr W M Pun for his assistance especially in matters concerning theory and computation.

I am most grateful to Mr Bob King for his assistance in setting up the experimental apparatus and in printing the excellent photographs presented in this thesis. Thanks are also due to Mrs Frith Oliver and Mrs Catherine Webb for their assistance in various administrative matters and to Mr Peter Dale for helping with computer problems.

The day-to-day exchange of ideas and informal discussion with fellow research workers have been most fruitful. I would like to thank my colleagues S B Jia, J Liu, S M Lo, F Villasenor, Z Y Wu and M Yamazaki and Dr S Farwagi for

making my stay at Imperial College an enjoyable one.

I would also like to thank Miss E M Archer and Miss L Handtor for providing very efficient library services.

I am very thankful to all my friends in London but particularly to Dr D W Gu. His words of encouragement helped me to overcome difficulties of various kinds.

I also wish to acknowledge the support by State Educational Commission of the People's Republic of China and Shanghai Jiao Tong University for maintenance and the Safety in Mines Research Establishment, Health and Safety Executive of Great Britain for financing the project.

Finally, my family has been very understanding and supportive throughout my study; I now plan to spend more time with my wife and daughter.

Xi Shi-Tong

席時桐

London

January 1986

<u>CONTENTS OF THESIS</u>	PAGE NO
ABSTRACT	3
PREFACE	5
CONTENTS OF THESIS	8
LIST OF FIGURES	14
CHAPTER 1 - INTRODUCTION	23
1.1 The problem considered	23
1.2 Practical relevance	25
1.3 Literature review on transient jets	26
1.31 Single-phase transient jets	26
1.32 Two-phase transient jets	32
1.4 Objectives of the present study	45
1.5 Outline of the thesis	47
CHAPTER 2 - THE MATHEMATICAL MODEL	49
2.1 Introduction	49
2.2 Definitions	50
2.21 Two fluids	50
2.22 Variables	51
2.3 Governing differential equations	52
2.4 Modelling assumptions	56
2.41 Constant-effective-viscosity model	56
2.42 Some assumptions of the two-fluid model	57
2.5 Empirical constants	60
2.6 Boundary and initial conditions	61
2.61 Boundary conditions	61

2.62	Initial conditions	61
2.7	Closure	62
CHAPTER 3 - THE NUMERICAL SOLUTION PROCEDURE		63
3.1	Introduction	63
3.2	The finite-difference grid	65
3.21	General concept	65
3.22	Location of flow variables	67
3.23	Control volume	69
3.3	The general finite-domain equation	70
3.4	Incorporation of boundary and initial conditions	75
3.41	Boundary conditions	75
3.42	Initial conditions	76
3.5	The "IPSA" solution procedure	77
3.6	Summary of the numerical solution procedure	78
CHAPTER 4 - COMPUTATION OF AN AIR-INTO-AIR STEADY JET		80
4.1	Introduction	80
4.2	The flow considered	82
4.3	Computational details	84
4.31	Solution procedure and calculation domain	84
4.32	The grid	86
4.33	Boundary conditions	90
4.34	Inlet boundary conditions	94
4.35	Similarity consideration	95

4.36	Convergence	95
4.37	Computer storage and computer time	98
4.4	Main results	100
4.5	Effect of empirical constants of two-fluid assumptions	104
4.6	Discussion	108
4.7	Conclusion	114
CHAPTER 5 - COMPUTATION OF AN AIR-INTO-AIR TRANSIENT JET		116
5.1	Introduction	116
5.2	The flow considered	118
5.3	Computational details	119
5.31	Solution procedure and calculation domain	119
5.32	The grid and time-step	121
5.33	Boundary conditions	127
5.34	Initial conditions	128
5.35	Convergence	128
5.36	Computer storage and computer time	130
5.4	Main results	132
5.5	Effect of empirical constants of two-fluid assumptions	136
5.6	Discussion	140
5.7	Conclusion	149
CHAPTER 6 - EXPERIMENTAL INVESTIGATION		151
6.1	Introduction	151
6.2	Apparatus	153

6.3	Experimental procedure	160
6.4	Experimental results	162
6.41	Experimental conditions	162
6.42	Results of brine-into-water and kerosene-into- water transient jets	166
6.43	Volume flow rate of injected fluids	172
6.44	Results of droplet-size measurement of kerosene-into-water steady jet	173
6.5	Discussion	185
6.6	Closure	191
CHAPTER 7 - COMPUTATION OF A BRINE-INTO-WATER TRANSIENT JET		193
7.1	Introduction	193
7.2	The flow considered	194
7.3	Computational details	195
7.31	Solution procedure and calculation domain	195
7.32	The grid and time-step	197
7.33	Boundary conditions	200
7.34	Initial conditions	202
7.35	Convergence	203
7.36	Computer storage and computer time	204
7.4	Definition of jet-tip	205
7.5	Computational results of tip-penetration rate	206
7.6	Effect of density ratio of injected brine to surrounding water	207

7.7	Discussion	212
7.8	Conclusion	214
CHAPTER 8 - COMPUTATION OF A KEROSENE-INTO-WATER TRANSIENT JET		216
8.1	Introduction	216
8.2	The flow considered	218
8.3	Modifications of two-fluid assumptions	219
8.4	Computational details	220
8.41	Solution procedure and calculation domain	220
8.42	The grid and time-step	220
8.43	Boundary conditions	223
8.44	Initial conditions	223
8.45	Convergence	224
8.46	Computer storage and computer time	225
8.5	Definition of the jet-tip	226
8.6	Computational results of tip-penetration rate	227
8.7	Effects of empirical constant C_f and density ratio of injected fluid to surrounding water	230
8.8	Discussion	230
8.9	Conclusion	239
CHAPTER 9 - CONCLUSIONS		241
9.1	Achievements and conclusions	241
9.11	Computational work	241
9.12	Experimental work	244
9.2	Suggestions for future work	245

9.21	Computational	245
9.22	Experimental	246
NOMENCLATURE		248
REFERENCES		251
APPENDIX A - The PHOENICS listings for the air- into-air transient jet		268
APPENDIX B - Listing of program "SPLINE"		274
APPENDIX C - Experimental data of number and mean diameter of droplets in cross-sections of kerosene-into- water steady jet		277
APPENDIX D - The PHOENICS listings for the kerosene- into-water transient jet		287

<u>LIST OF FIGURES</u>	PAGE NO
1.1-1 A transient jet	25
3.21-1 A two-dimensional cartesian control cell	66
3.22-1 Grid showing location of variables and their control volumes	68
4.2-1 A round air-into-air jet	83
4.31-1 The calculation domain for the air-into- air steady jet	85
4.32-1 Effect of radial grid-size on the mean velocity along the centre-line	88
4.32-2 Effect of radial grid-size on the predicted similarity-profiles of the streamwise velocities	88
4.32-3 Effect of radial grid-size on the predicted similarity-profile of the intermittency factor	89
4.32-4 Effect of forward step-size on the mean velocity along the centre-line	91
4.32-5 Effect of forward step-size on the predicted similarity-profiles of the streamwise velocities	91
4 32-6 Effect of forward step-size on the predicted similarity-profile of the intermittency factor	92
4.33-1 The boundaries of the calculation domain	93

	for the air-into-air steady jet	
4.35-1	Test for similarity on the predicted profiles of the streamwise velocities	96
4.35-2	Test for similarity on the predicted profile of the intermittency factor	96
4.4-1	Mean velocity along the centre-line	101
4.4-2	Similarity-profiles of the streamwise velocities	101
4.4-3	Similarity-profile of the intermittency factor	103
4.5-1	Effect of C_m on the mean velocity along the centre-line	106
4.5-2	Effect of C_m on the predicted similarity-profiles of the streamwise velocities	106
4.5-3	Effect of C_m on the predicted similarity-profile of the intermittency factor	107
4.5-4	Effect of C_f on the mean velocity along the centre-line	109
4.5-5	Effect of C_f on the predicted similarity-profiles of the streamwise velocities	109
4.5-6	Effect of C_f on the predicted similarity-profile of the intermittency factor	110
4.5-7	Effect of C_d on the mean velocity along the centre-line	111
4.5-8	Effect of C_d on the predicted similarity-profiles of the streamwise velocities	111
4.5-9	Effect of C_d on the predicted similarity-	112

	profile of the intermittency factor	
5.31-1	The calculation domain for the air-into-air transient jet	120
5.32-1	Effect of grid-node number on mean velocity along the centre-line	124
5.32-2	Effect of grid-node number on mean velocity at given axial stations	124
5.32-3	Effect of time-step on mean velocity along the centre-line	125
5.32-4	Effect of time-step on mean velocity at given axial stations	125
5.33-1	The boundaries of the calculation domain for the air-into-air transient jet	126
5.4-1	Variation of mean velocity along the centre-line with time	133
5.4-2	Development of mean velocity at given axial stations: case 5-1	133
5.4-3	Development of mean velocity at given axial stations: case 5-2	135
5.4-4	Development of mean velocity at given axial stations: case 5-3	135
5.4-5	Predicted profiles of streamwise velocities	137
5.4-6	Predicted profiles of intermittency factor	138
5.4-7	Predicted profiles of streamwise velocities	139
5.4-8	Predicted profiles of intermittency	139

	factor	
5.5-1	Effect of C_m on mean velocity along the centre-line	141
5.5-2	Effect of C_m on mean velocity at given axial stations	141
5.5-3	Effect of C_m on the predicted profiles of streamwise velocities	142
5.5-4	Effect of C_m on the predicted profiles of intermittency factor	142
5.5-5	Effect of C_f on mean velocity along the centre-line	143
5.5-6	Effect of C_f on mean velocity at given axial stations	143
5.5-7	Effect of C_f on the predicted profiles of streamwise velocities	144
5.5-8	Effect of C_f on the predicted profiles of intermittency factor	144
5.5-9	Effect of C_d on mean velocity along the centre-line	145
5.5-10	Effect of C_d on mean velocity at given axial stations	145
5.5-11	Effect of C_d on the predicted profiles of streamwise velocities	146
5.5-12	Effect of C_d on the predicted profiles of intermittency factor	146
6.2-1	Schematic representation of the general lay-out of the experimental apparatus	154

6.2-2	Overall view of apparatus	155
6.2-3	Dimensions of the main tank	156
6.42-1	Flow patterns of a brine-into-water and a kerosene-into-water transient jets	167
6.42-2	Experimental data of the tip-penetration of a brine-into-water transient jet: case 6-1	168
6.42-3	Experimental data of the tip-penetration of a brine-into-water transient jet: case 6-2	168
6.42-4	Experimental data of the tip-penetration of a brine-into-water transient jet: case 6-3	169
6.42-5	Experimental data of the tip-penetration of a brine-into-water transient jet: case 6-4	169
6.42-6	Experimental data of the tip-penetration of a kerosene-into-water transient jet: case 6-5	170
6.42-7	Experimental data of the tip-penetration of a kerosene-into-water transient jet: case 6-6	170
6.42-8	Experimental data of the tip-penetration of a kerosene-into-water transient jet: case 6-7	171
6.42-9	Experimental data of the tip-penetration of a kerosene-into-water transient jet: case 6-8	171

6.43-1	Experimental data of the volume flow rate at the exit of brine-into-water transient jets: case 6-1,3	174
6.43-2	Experimental data of the volume flow rate at the exit of brine-into-water transient jets: case 6-2,4	174
6.43-3	Experimental data of the volume flow rate at the exit of a kerosene-into-water transient jet: case 6-5	175
6.43-4	Experimental data of the volume flow rate at the exit of a kerosene-into-water transient jet: case 6-6	175
6.43-5	Experimental data of the volume flow rate at the exit of a kerosene-into-water transient jet: case 6-7	176
6.43-6	Experimental data of the volume flow rate at the exit of a kerosene-into-water transient jet: case 6-8	176
6.44-1	Overall views of kerosene-into-water steady jet: cases 6-9, 10, 11	177
6.44-2	Overall views of kerosene-into-water steady jet: cases 6-12, 13, 14	178
6.44-3	Overall views of kerosene-into-water steady jet: cases 6-15, 16, 17	179
6.44-4	Volume percentage of droplets in diameter-size groups in a cross- section: $Z=50D_n-60D_n$	181

6.44-5	Experimental data of the axial distributions of the mean droplet diameters of kerosene-into-water steady jets	186
6.44-6	Experimental data of the mean droplet diameters of kerosene-into-water steady jets	187
7.31-1	The calculation domain for the brine- into-water transient jet	197
7.32-1	Effect of grid-node number on the height of the jet-tip	199
7.32-2	Effect of time-step on the height of the jet-tip	199
7.33-1	The boundaries of the calculation domain for the brine-into-water transient jet	201
7.5-1	Tip-penetration rate of a brine-into- water transient jet: case 7-1	208
7.5-2	Tip-penetration rate of a brine-into- water transient jet: case 7-2	208
7.5-3	Tip-penetration rate of a brine-into- water transient jet: case 7-3	209
7.5-4	Tip-penetration rate of a brine-into- water transient jet: case 7-4	209
7.6-1	Effect of ρ_1 / ρ_2 on the predicted tip- penetration rate of a brine-into-water transient jet	210
7.6-2	Effect of ρ_1 / ρ_2 on the predicted profiles of streamwise velocities	211

7.6-3	Effect of ρ_1 / ρ_2 on the predicted profile of intermittency factor	211
8.42-1	Effect of grid-node number on the height of the jet-tip	222
8.42-2	Effect of time-step on the height of the jet-tip	222
8.6-1	Tip-penetration rate of a kerosene-into-water transient jet: case 8-1	228
8.6-2	Tip-penetration rate of a kerosene-into-water transient jet: case 8-2	228
8.6-3	Tip-penetration rate of a kerosene-into-water transient jet: case 8-3	229
8.6-4	Tip-penetration rate of a kerosene-into-water transient jet: case 8-4	229
8.6-5	Predicted profiles of streamwise velocities	231
8.6-6	Predicted profiles of volume fraction of the kerosene	232
8.7-1	Effect of C_f on the predicted tip-penetration rate of a kerosene-into-water transient jet	233
8.7-2	Effect of C_f on the predicted profiles of streamwise velocities	234
8.7-3	Effect of C_f on the predicted profile of r_1	234
8.7-4	Effect of ρ_1 / ρ_2 on the predicted tip-penetration rate of a kerosene-into-	235

water transient jet

8.7-5	Effect of ρ_1 / ρ_2 on the predicted profiles of streamwise velocities	236
8.7-6	Effect of ρ_1 / ρ_2 on the predicted profile of r_1	236

CHAPTER 1 - INTRODUCTION

1.1 The problem considered

Transient, turbulent, round jets of miscible and immiscible fluids are considered in the present study.

A transient jet occurs when a fluid is injected through a nozzle into a surrounding fluid which may be flowing or stagnant.

The injected fluid, which is miscible with the surrounding fluid, forms a single-phase flow situation, while that which is immiscible with the surrounding fluid forms a two-phase flow situation.

Three transient, turbulent, round jets are considered in the present study, namely, air is injected into air; brine into tap water; and kerosene into tap water. In all these cases, the surrounding fluid is initially at rest. These jets will be referred to as "air-into-air transient jet", "brine-into-water transient jet", and "kerosene-into-water transient jet". The "air-into-air" jet is also considered under steady-state conditions; this will be referred to as "air-into-air steady jet".

The air-into-air steady, or transient jets and the brine-into-water transient jet are single-phase flows while the

kerosene-into-water jets are two-phase flows. The brine-into-water and kerosene-into-water jets are affected by the buoyancy force due to the density difference between the injected and surrounding fluids.

In order to explain the phenomenon under study, the air-into-air transient jet is detailed in the following paragraph:

Immediately after opening a valve, the reservoir air injects into and mixes with the surrounding air, setting the latter in motion. If the reservoir pressure is not too high, a sub-sonic transient jet will be formed as shown in Fig.1.1-1. As the jet penetrates further, it entrains more of the surrounding air resulting in a developing jet. The duration of such a jet depends on, among other factors, the capacity of the reservoir; if the reservoir capacity is infinitely large compared with the discharge rate, the jet would persist infinitely. At any given point inside the jet field, a time would be reached when the flow there achieves a steady-state. The time required for this steady-state to be reached depends on the location of the point; the further the point away from the exit is, the longer the time is.

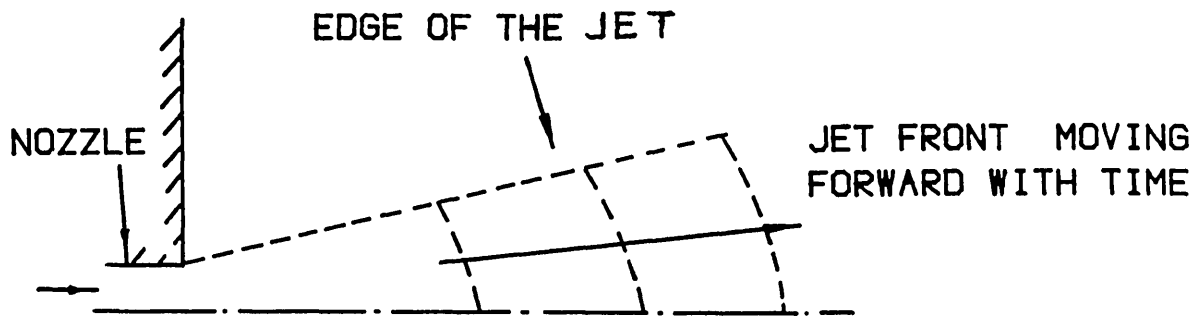


FIG. 1.1-1 A TRANSIENT JET

Computations and experiments relating to such transient turbulent jets with various flow conditions are carried out in the present study.

1.2 Practical relevance

Transient jets are important in many engineering processes such as:

(1) High pressure containers: gases may accidentally leak from high pressure containers during transportation or production. A fire hazard would result if the gases are combustible.

(2) Compression-ignition (Diesel) engines: fuels are injected into the combustion chamber prior to ignition.

(3) Rocket engines: gases exhaust from the rocket nozzle during the initial build-up of thrust, and a secondary flame may occur when the exhaust gases mix with the surrounding air.

(4) Flame traps and flame-proof enclosures: hot gases are ejected from the flange gap of flame traps or flame-proof enclosures.

In all these practical problems, understanding behaviours of the transient jet prior to its achieving a steady-state is important, since it will improve predictions of the performance of such engineering devices. Thus, the designers would want to know, not only qualitatively, but also quantitatively, the influences of various parameters and the way in which they affect the flow characteristics.

1.3 Literature review on transient jets

The literature on transient jets is now reviewed; it is presented in two parts which relate to single-phase and two-phase transient jets. Theoretical and experimental investigations are described.

1.31 Single-phase transient jets

(1) Theoretical investigations

Bhuta (1961) obtained an analytical solution for an incompressible, unsteady, two-dimensional, laminar or turbulent jet. A perturbation method was employed in the study to linearise the Prandtl boundary-layer equation. For the same problems, a numerical solution was obtained by Kent (1972) based on the integral method. This method assumes the similarity in the flow nature and ignores the effect of the pressure gradient in the flow field. Ma and Ong (1971) and Wirz (1973) have also made use of the similarity assumption. Wirz studied a compressible, two-dimensional, unsteady jet mixing and again restricted the investigation to boundary layer flows.

Payne (1958) first presented a solution of the two-dimensional unsteady equations for a plane starting-jet at a low Reynolds number. Starting with Helmholtz's vorticity equation, Payne employed a step-by-step time-interval method. In this method, a central finite-difference formulation was used to calculate the time and spatial derivatives. The vorticity distribution and streamlines were obtained at successive time steps for an incompressible, turbulent, round, starting-jet.

Ma and Ong (1971) replaced the central finite-difference scheme with the upwind difference scheme and extended the investigation to cover high Reynolds numbers. Different to Payne's approach, they used the similarity variables to

transform the three-dimensional governing equations into two-dimensional ones; these were then solved by the Gauss-Seidel iterative procedure. Although they satisfactorily predicted the experimental data reported in the same paper, the "similarity" method is limited to problems which possess similar-solutions. In the case of a transient jet, the similar-solution can only be valid at large axial distances downstream of the nozzle.

For transient flows with temperature gradients and chemical reactions, Phillips (1971) carried out a theoretical study of the suppression of explosions by narrow gaps. The ignition of the combustible mixtures was assumed to occur in a spherical vortex at the head of the turbulent transient jet. A one-dimensional energy balance for a small gas pocket inside the vortex was then performed. The maximum safety gap was obtained from solving the resulting ordinary differential equation. This method, however, can not be employed for the detailed study of a phenomenon which is two-dimensional and time dependent in nature.

Hengrussamee (1975) developed a calculation procedure capable of predicting the mixing of hot, turbulent, round, transient jets with either inflammable or chemically inert surroundings. The investigation involved the formulation of turbulence and chemical reaction models to represent the physical situation and the use of a velocity-pressure

numerical approach to solve the resulting unsteady two-dimensional governing equations. The $k-\epsilon$ model proposed by Launder et al. (1972) was used in Hengrussamee's calculations. A set of model constants, appropriate to plane jets and mixing layers, was adopted in the $k-\epsilon$ model. The theoretical predictions of the jet front position were satisfactory in comparison with Phillips' experimental data for both non-ignition and ignition cases. However, the predicted jet penetration rates were higher than Ong's experimental values (1970).

Scaling laws for incompressible, turbulent, as well as laminar, round, starting-jets were deduced by Kuo (1982). The definition was adopted that a jet reached steady-state between the nozzle and an axial location when, at that location, the centreline velocity achieves 70% of its steady-state value. Characteristic steadying length and time scales, as well as a unique function that relates dimensionless steadying time to dimensionless steadying length were determined. The time and length scales and the functions are different for turbulent and laminar starting-jets. For the turbulent starting-jet, the numerical solution of the Navier-Stokes equations was obtained using a modified $k-\epsilon$ model of turbulence. The constants of the $k-\epsilon$ model were optimized by comparing centreline velocity, mean radial velocity and streamwise kinetic energy distributions with those measured by other

authors in steady round jets. Parametric studies of the starting-jet were carried out for a range of Reynolds number.

(2) Experimental investigations

Early experiments on single-phase transient jets were confined to observations of the flow patterns resulting from the mixing of isothermal homogeneous fluids. Rizk (1955) performed experiments to visualise the development and mixing of round, water, starting-jets in conjunction with the scavenging of two-stroke engines. Smoke-tracers were used by Meyer (1961) to follow the mixing of a low-velocity air jet with stagnant surroundings. These two studies, however, were qualitative in nature; the instantaneous velocities of the flow field were not measured quantitatively.

The first quantitative detailed experimental investigation was reported by Ma and Ong (1971). They carried out experiments for a turbulent, round, air-into-air starting-jet. The streamwise velocity at several axial stations downstream of the nozzle was measured using a hot-wire anemometer.

To obtain the maximum safe-gap size which prevents the flame transmission from the inside of an enclosure into inflammable surroundings, Wolfhard and Bruszk (1959)

and Phillips (1971) investigated transient jets with non-uniform fields of temperature and concentration. In Phillips' experiments, ignition or non-ignition phenomena were achieved by adjusting the composition of the mixing media. In both cases, Schlieren photographs were taken for the developing hot, sub-sonic transient jet emerging from a small nozzle on an explosion cylinder with large pressure release vents. Wolfhard and Bruszk used a sealed explosion-cylinder placed in an inflammable atmosphere which was enclosed inside a large duct.

Abramovich and Solan (1973) presented experimental data for the speed of travel of the spherical vortex at the front of incompressible, round, impulsively started, laminar jets. It was found that, for a given exit velocity profile, the time required for the centreline velocity of the jet to reach 70% of its steady-state value at any axial location was a function of the diameter of the orifice, the mean exit velocity, and the kinematic viscosity. They proposed a characteristic axial length and time scale.

For an impulsively started air-into-air jet, Witze (1980) measured, using hot-film anemometry, the steady and transient streamwise velocities along the centre-line. Since this study related to fuel-injected internal combustion engines, two nozzles with complex

configurations were used in the experiments.

Demetriou (1983) obtained experimental results concerning the geometry and streamwise velocity of a homogeneous, round, water, starting-jet. The results were compared with a homogeneous steady-jet and a starting-plume. A qualitative description of the motion of the starting-jet front was given by dividing the jet trajectory into three regions. Beyond twenty nozzle diameters, the jet front has a constant character and a streamwise velocity constantly less than that of the steady-jet; in the range less than nine nozzle diameters, the starting-jet front accelerates; and in the intermediate region between them, it has a transient behaviour.

1.32 Two-phase transient jets

Due to the scope of the present study, this part of the review is mainly concerned with the penetration and droplet-size distribution of fuel sprays without evaporation and combustion; some studies of two-phase steady jets are also included.

(1) Theoretical investigations

Lord Rayleigh (1878, 1892) first theoretically investigated the instability of a jet under viscous and capillary forces and established the following criterion:

if the wavelength of a disturbance approaches 4.508 times the diameter of a cylindrical jet, the jet becomes unstable and breaks into droplets. A similar investigation was carried out by Weber (1931) who studied the instability under viscous, capillary and aerodynamic forces. However, both studies were made for jets with extremely low velocities.

Based upon experiments or theoretical analyses with some assumptions, many expressions have been developed for calculating the spray penetration in Diesel engines. Contributing researches include those of Schweitzer (1937), Lyshevskiy (1955, 1956), Wakuri et al. (1960), Rusinov (1963), Sitkei (1964), Parks et al. (1966), Ogasawara and Sami (1967), Öz (1969), Burt and Troth (1970), Taylor and Walsham (1970), and Dent (1971). Hay and Johnes (1972) examined these expressions and recommended the use of the Dent correlation under all conditions except for large chamber densities where the Dent correlation tends to overpredict the spray penetration. The correlations of Schweitzer, Wakuri et al. and Dent show that the spray penetration is proportional to the square root of time.

Difficulties in adequately simulating two-phase turbulent flows in sprays have inhibited the development of spray models. Nevertheless, even oversimplified models have

shown some success and contributed to the understanding of spray processes. A selection of such models is outlined below.

One-dimensional models provide the means of considering interactions between the liquid and gas phases, while avoiding the complexities of drop diffusion in a turbulent gas flow.

Liquid rocket engine is an area where one-dimensional theoretical models of spray combustion are widely accepted. Spalding (1959) presented an early analysis of this type. Williams (1965) , Priem and Heidmann (1960) , Burstein et al. (1962), Sutton (1970) also developed one-dimensional models for the rocket engines.

Bracco (1973) solved the transient, one-dimensional spray equation to simulate drop evaporation in a Wankel engine. The energy equation for the gas phase was solved, and the gas velocity was computed allowing for fuel evaporation but neglecting spatial pressure gradients.

The one-dimensional models mentioned above, take an oversimplified look at the processes taking place and involve a substantial amount of empiricism. Experience with these models has led to an optimized selection of physical properties and empirical parameters. Their success has been helpful in design; but this in no way

implies that they are satisfactory in all respects.

Models, which attempt to describe the detailed structure of the turbulent two- and three-dimensional spray processes have become attractive especially those relating to furnaces, gas turbines, Diesel engines, etc.. In such models, two major categories can be distinguished, namely, Locally Homogeneous Flow (LHF) models and Separated Flow (SF) models. In the LHF models, the liquid and gas phases are assumed to be in dynamic and thermodynamic equilibrium, i.e. at each point in the flow both phases have the same velocities and temperature. On the other hand, SF models consider the effect of finite rates of transport between the phases.

Early LHF models by Thring and Newby (1953), Newman and Brzustowski (1971) and Shearer and Faeth (1977) employed integral models to represent the flow. Although these simplified models were calibrated to only a limited degree with more basic and well-defined flows, they achieved broad agreement with overall phenomena such as flame length and spray boundaries.

Later LHF models by Khalil and Whitelaw (1976), Mao et al. (1981) employed higher-order turbulence models which were calibrated using measurements in gaseous combusting and non-combusting flows. By comparison with detailed

measurements of spray structure, the LHF models were found to invariably overestimate the rate of development of the flow, this is due to the neglect of finite interphase transport rates.

The LHF models, however, can only represent cases in which a spray consists of infinite droplets. Models which take the effects of transport processes between the phases into account are generally needed for quantitative predictions of practical sprays.

The separated flow (SF) models of multiphase flow specifically treat the finite rate of exchange of mass, momentum, and energy between the phases. Current SF models generally average over processes, which occur on a scale comparable to the droplet size, i.e. no attempt is made to accurately model the details of the flow field around or within individual drops; this is due to the limitation of computer storage and computation costs. Therefore, the exchange processes between the phases must be modeled independently, usually by employing empirical expressions for droplet drag and for heat and mass transfer rates.

Most separated flow analyses of sprays employ an approach identified as the "particle-source-in-cell" (PSIC) or "discrete droplet model" (DDM). This involves dividing the spray into representative samples of discrete drops

whose motion and transport through the flow field are treated by Lagrangian formulation. A Eulerian formulation, similar to that used in LHF models, is employed to solve the governing equations for the gas phase. The effects of drops on the gas phase are modeled by introducing appropriate source terms in the gas phase motion equation.

Examples giving typical features of current DDM models appear in the work of Crowe (1974, 1978) , Alpert and Mathews (1979), Mongia and Smith (1978) , Butler et al. (1980), EL Banhawy and Whitelaw (1980) , Gosman and Johns (1980), and Swithenbank et al. (1976,1980).

Different to DDM, the "continuous droplet model" (CDM) approach describes drop properties using a continuous statistical distribution function at all positions in the flow field. The development of this method for sprays was due to Williams (1965). To date, CDMs have been limited to cases where the drops of pure liquid are evaporating at a steady wet-bulb condition.

Past applications of the CDM employed a relatively simplified representation of spray phenomena; this can be found in the work of Bracco (1973) and Westbrook (1977).

The third approach to separated flow analysis of sprays is

the continuum formulation model (CFM) . There has been a substantial amount of progress in modelling multidimensional two-phase flows using CFM including the work of Harlow and Amsden (1975a, 1975b) and Dehaye et al. (1981).

In most of the early work on separated flow models, the sprays were assumed to be dilute, the turbulent dispersion of drops was ignored and the influence of dynamic effects and turbulence on drop transport rates was neglected.

For dilute sprays, two main methods have been employed to compute particle dispersion in sprays. The earliest investigation modelled turbulent dispersion using a gradient diffusion expression which employed a turbulent gas/particle exchange coefficient [Jurewicz et al., 1976, 1977].

Stochastic methods are the most recent development in modelling particle dispersion in sprays. With stochastic particle dispersion models, several turbulence models have been used in the computations of Buckingham and Siekhaus (1979, 1981) and Peskin and Kau (1979).

The dense spray region is characterized by discrete drops in a more-or-less continuous gas phase. Within the dense spray region, the interaction between drops is important; this implies that the effects of, drop spacing on

transport rates, drop collisions, modifications of turbulence properties due to the presence of drops, and the volume occupied by the liquid phase, all become significant.

The model reported by O'Rourke and Bracco (1980) represents one of the first comprehensive treatments of dense sprays. They used a discrete droplet model (DDM) with a Lagrangian treatment of the liquid phase. Turbulence was represented by a constant eddy diffusivity together with a turbulent Prandtl/Schmidt number of unity. The effect of turbulent fluctuations on the drag and heat and mass transfer properties of drops was considered using a stochastic model. However, turbulent drop dispersion was neglected. The prediction obtained by O'Rourke and Bracco indicated that the effects of adjacent drops on drop transport rates are relatively small for void fractions greater than 0.9; however, drop collision effects are still significant at relatively high void fractions.

Kuo (1982) deduced a scaling law for transient and steady, non-evaporating and evaporating sprays through parametric studies with a set of coupled two-dimensional unsteady conservation equations for the gas and the liquid. A unique function was determined for the penetration rate of an impulsively started thick spray.

There is a pressing need to overcome the experimental difficulties in the dense spray region so that further progress can be made towards developing reliable prediction for this region.

(2) Experimental investigations

- Spray penetration

The characteristics of spray penetration have been studied since the early days of the development of the Diesel engine, with contributions from workers in Germany, the U.S.A., and the U.K..

The early experimental investigations on the spray penetration in Diesel engines were carried out by Holfelder (1932) , Mehlig (1934) , and Schweitzer (1937). Mehlig deduced a relationship between the streamwise velocity and backpressure from theoretical analysis and experiment. Schweitzer extended the relationship to a comprehensive form of spray penetration. Schweitzer had also found that increasing fuel viscosity increased spray penetration.

The experiments on the engine spray penetration performed by Gray et al. (1966) , and Taylor (1967), had shown that the spray penetration of Schweitzer was applicable to a variety of cold bomb studies.

However, from the extensive hot bomb studies of Parks et al. (1966), it was found that Schweitzer's relationship for the penetration was not applicable to the hot bomb studies, as it did not allow for temperature effects.

Parks et al. put forward a correlation of the spray penetration based on their experimental data.

Taylor (1967) carried out penetration studies on an engine (8 in. bore, 245 lbf/in bmeP, 750 rpm) . Applying dimensional analysis, he obtained a correlation from his experimental data. This showed a deviation from Schweitzer's relationship.

The effect of fuel properties on the spray tip-penetration was the main aspect of the experimental investigation by Burt and Troth (1970) . The spray measurements were carried out on fuels with a wide range of viscosities including a gasoline, a kerosene, and a heavy gas oil. Values of the spray tip-penetration were obtained for a range of pressures in a chamber which was filled with nitrogen and helium.

The experimental studies of Rife and Heywood (1974) , Reitz (1978) , Arai et al. (1985) , and Yule et al. (1985), focused particular attention on the initial stages of injection.

Rife and Heywood compared experimental data on the geometry of a jet in a quiescent combustion chamber with results from a two-phase jet model. They concluded that the centreline velocity of the fuel spray at the start of injection is constant.

Based on the extensive photographic study of jet atomization, Reitz also reported that the velocity of the fuel jet upon injection is constant.

Arai et al. measured the break-up length and spray angle of a high speed jet under high ambient pressure conditions. The effects of the nozzle hole length on the break-up length and the spray angle were studied by considering the break-up mechanism of the high speed liquid jet.

Yule et al. made an extensive set of measurements for the pulsed, non-burning sprays produced by three single hole Diesel injectors. Penetration rates and spray angle measurements were made in a quiescent chamber and, with cross-flow, in a wind tunnel. The pressure and temperature of the gas were also varied.

- Droplet-size distribution in sprays

Hiroyasu and Kadota (1974) presented time-averaged values of droplet size. In their experiment, Diesel fuel oil was injected into a high pressure, low-temperature gaseous environment using a Diesel-type injection system. The droplet size was measured using the liquid sampling technique. The ambient pressure, pump speed, and rack position of the fuel pump were varied and their effect on droplet size was studied.

Simmons (1977) accumulated and presented data on the drop-size/volume-fraction distribution of sprays observed with a large number of gas-turbine nozzles of different types. The data included measurements for a range of fuel viscosities, at a variety of operating conditions. Both optical and wax-droplet methods were used.

The droplet number density and size distribution of a solid-cone fuel spray were measured by Tishkoff et al. (1980) . A commercially available Malvern ST-1800 spray analyzer was used in the experiment.

During the injection period, the injection pressure, the nozzle orifice area, and the injection rate, fluctuate. Consequently, the droplet sizes change with time during this period. Hiroyasu et al. (1978) measured the transient characteristics of droplet size. They presented the variation of the frequency-distribution of the droplet volume with sampling time.

Saito et al. (1985) conducted experiments using the swirl nozzle of a Diesel engine. The immersion method was used to measure the drop-size distribution under high pressures within the injection bomb. They found that the effect of the sampling location on the drop-size distribution and the mean drop diameter was rather smaller than that of the hole type nozzle.

Recently, the diameter of drops at the edge of a spray in the immediate vicinity of the nozzle exit was measured by Wu et al. (1985). They used a photographic technique with back illumination. The average diameter was given for various spray conditions. The measured diameter showed anticipated change when the injection velocity, liquid properties and nozzle geometry were varied. However, when the gas density was varied, the expected change in the average diameter was not obtained.

Finally, for dilute sprays, numerous optical techniques have been used to measure drop sizes in the range of $10\text{ }\mu\text{m}$ to $40\text{ }\mu\text{m}$. These include the imaging techniques of McCreath et al. (1972) and Takeuchi et al. (1983); diffraction techniques of Swithenbank et al. (1976), and Ruscello and Hirleman (1981); techniques based on Mie scattering of Jones (1979) and Kerker (1969); interferometric or visibility techniques of Allano et al. (1982) and Hishida et al. (1982).

1.4 Objectives of the present study

The overall objective of the present study is to investigate the behaviour of miscible and immiscible, transient, turbulent jets using computational and experimental methods.

The particular objectives are:-

- (1) To simulate numerically turbulent, round, air-into-air steady and transient jets, and to compare the computational results with available experimental data;
- (2) To observe experimentally, the similarities between miscible (brine-into-water) and immiscible (kerosene-into-water), transient, turbulent jets;
- (3) To measure the tip-penetration rates of the two transient jets in (2) above.
- (4) To measure the droplet size of a kerosene-into-water jet under steady-state conditions; and
- (5) To predict the tip-penetration rates of the two transient jets, and to compare the predictions with the experimental results of the present work.

As far as the mathematical model is concerned, the present study pays special attention to testing the suitability of the constant-effective-viscosity assumption to both steady and transient, turbulent jets. The two-fluid assumptions are tested only in respect of the intermittency phenomenon because the constant-effective-viscosity assumption masks effects of the two-fluid assumptions on the mean flow. The present study also attempts to explore the possibility to treat both miscible and immiscible, turbulent flows using same modelling assumptions.

1.5 Outline of the thesis

The main body of the thesis consists of nine chapters of which this Introduction is the first.

Chapter 2 gives the mathematical model used. It describes the partial-differential equations governing the flows under consideration and the modelling assumptions made. The empirical constants used in the present study are given together with the boundary and initial conditions.

Chapter 3 describes the numerical solution procedure. The partial-differential equations are reduced to the finite-domain equations by integration over small computational cells.

The simulations of the turbulent, round, air-into-air steady and transient jets are presented in Chapters 4 and 5 respectively. The computational results are compared with available experimental data.

Chapter 6 is concerned with the present experimental work. Similarities between miscible and immiscible transient, turbulent jets are observed. These experiments provide data for testing the numerical predictions. They also supply the necessary information for boundary conditions and modelling assumptions to be used in the computations.

Chapter 7 presents the computer simulation of the brine-into-water transient jet while Chapter 8 gives that of the kerosene-into-water transient jet. The predicted tip-penetration rates of the two jets are compared with the present experimental data. The effects of the density ratio of the injected fluid to the surrounding fluid are also demonstrated.

Finally, Chapter 9 summarizes the main conclusions of the present study and makes some suggestions for future research. This chapter is followed by Nomenclature and References, along with Appendices which give supplementary information concerning the work presented in this thesis.

CHAPTER 2 - THE MATHEMATICAL MODEL

2.1 Introduction

The first step to analyse a fluid-dynamics problem is to provide the mathematical model based on the physical laws governing the flow considered. This chapter presents the model for the present work.

In the present study, the constant-effective-viscosity model [Schlichting, 1966, Spalding, 1979] is used to calculate the turbulent viscosity. In addition to the constant-effective-viscosity assumption, some assumptions of the two-fluid model [Spalding, 1982a], which will be referred to as "two-fluid assumptions", are incorporated into the mathematical model; and they are tested in respect of the intermittency phenomenon in turbulent flows.

Different from conventional mathematical models, the present one treats both single-, and two-phase turbulent flows as random interspersions of two fluids each of which possesses its own velocity components, temperature, composition etc., and interacts through the sharing of space and exchange of mass, momentum, and energy. Therefore, two sets of differential equations are required to describe the motion of the two fluids. These differential equations, the modelling assumptions and the

boundary and initial conditions compose a complete mathematical model.

Both miscible and immiscible flows are considered in the present study. This chapter gives the general conservation equations for both flows, and describes the modelling assumptions used in the miscible flow cases. For the immiscible flow, the same constant-effective-viscosity model is used; but the two-fluid assumptions are modified. These modifications will be given in Chapter 8.

The outline of the present chapter is as follows:

Section 2.2 gives the definitions of the two fluids and relevant variables while Section 2.3 presents the partial-differential equations which govern the flows under consideration. Section 2.4 describes the modelling assumptions, followed by a description of the empirical constants used in these modelling assumptions. Finally, Section 2.6 describes the boundary and initial conditions.

2.2 Definitions

2.21 Two fluids

It is supposed that two fluids, fluid-1 and fluid-2, share occupancy of space. The proportions of time during which

each can be expected to occupy a particular location are called "volume fraction" , r_1 and r_2 . These two volume fractions obey the relation:

$$r_1 + r_2 = 1 \quad (2.21-1)$$

For the miscible flows of the present study, fluid-1 and fluid-2 are defined as "turbulent" (rotational) and "non-turbulent" (irrotational) fluids. Consequently, the volume fraction of the turbulent fluid, i.e. that of fluid-1, is interpreted as the intermittency factor I [Spalding, 1984a].

2.22 Variables

- Independent variables

The present study is concerned with steady or transient two-dimensional, round jets. The polar-coordinate system is employed. Therefore, the independent variables are:

y - radial coordinate;
 z - axial coordinate; and
 t - time.

Obviously, for a steady problem, the " t " becomes irrelevant.

- Dependent variables

The main dependent variables are:

- r_i - the volume fraction;
- V_i - the local average radial velocity;
- W_i - the local average streamwise velocity; and
- P - the pressure.

Subscript i is equal to 1 or 2 which denotes fluid-1 or fluid-2. The pressure is supposed to be shared by the two fluids and therefore, has no subscript.

Time-average values of the mean flow are obtained from those of the two fluids via the following relation:

$$\phi = r_1\phi_1 + r_2\phi_2 \quad (2.22-1)$$

where ϕ denotes the velocity component, V or W .

Thus, there are seven dependent variables. The present task aims to predict, for each of these variables, its distribution in two space dimensions; also a time dimension is added when a transient flow situation is considered.

2.3 Governing differential equations

The differential equations governing fluid-dynamics phenomena are the conservation equations which balance specific properties (physical quantities) of the flow.

Here, these equations will not be derived from first principles, for they may be found elsewhere (Spalding, 1982a).

(1) Conservation of mass

For fluid-1

$$\begin{aligned} \frac{\partial}{\partial t} (\rho_1 r_1) + \frac{\partial}{\partial z} (\rho_1 r_1 W_1) + \frac{1}{y} \frac{\partial}{\partial y} (y \rho_1 r_1 V_1) = \\ \frac{1}{y} \frac{\partial}{\partial y} (y \rho_1 D \frac{\partial r_1}{\partial y}) + m \end{aligned} \quad (2.3-1)$$

For fluid-2

$$\begin{aligned} \frac{\partial}{\partial t} (\rho_2 r_2) + \frac{\partial}{\partial z} (\rho_2 r_2 W_2) + \frac{1}{y} \frac{\partial}{\partial y} (y \rho_2 r_2 V_2) = \\ \frac{1}{y} \frac{\partial}{\partial y} (y \rho_2 D \frac{\partial r_2}{\partial y}) - m \end{aligned} \quad (2.3-2)$$

where m represents the rate of transfer of mass from fluid-2 to fluid-1.

The gradient-diffusion term on the RHS of each of the above equations, is associated with the instantaneous departure from the mean velocity, resulting from fluctuating vorticity; D represents the gradient-diffusion coefficient which will be given in Section 2.4 below.

(2) Conservation of momentum

The momentum conservation equations for the two fluids can be expressed in the following general form:

$$\begin{aligned} \frac{\partial}{\partial t} (\rho_i r_i \phi_i) + \frac{\partial}{\partial z} (\rho_i r_i W_i \phi_i) + \frac{1}{y} \frac{\partial}{\partial y} (y \rho_i r_i V_i \phi_i) = \\ \frac{\partial}{\partial z} (r_i \mu_i \frac{\partial \phi_i}{\partial z}) + \frac{1}{y} \frac{\partial}{\partial y} (y r_i \mu_i \frac{\partial \phi_i}{\partial y}) + \frac{1}{y} \frac{\partial}{\partial y} (y \rho_i D \phi_i \frac{\partial r_i}{\partial y}) + S_{\phi_i} \quad (2.3-3) \end{aligned}$$

where $i=1$ for fluid-1;

$i=2$ for fluid-2;

ϕ_i is the velocity component, V or W ;

μ_i is the turbulent or laminar viscosity within one fluid; and

S_{ϕ_i} is the source of the variable, as shown in Table 2.3-1 below.

Variable, ϕ_i	Source, $S_{\phi i}$
V_1	$-r_1 \left(\frac{\partial P}{\partial Y} + B_{1Y} \rho_1 \right) + V_1 m - F_V$
V_2	$-r_2 \left(\frac{\partial P}{\partial Y} + B_{2Y} \rho_2 \right) - V_2 m + F_V$
W_1	$-r_1 \left(\frac{\partial P}{\partial Z} + B_{1Z} \rho_1 \right) + W_1 m - F_W$
W_2	$-r_2 \left(\frac{\partial P}{\partial Z} + B_{2Z} \rho_2 \right) - W_2 m + F_W$

Table 2.3-1 Source terms in momentum equation

Table 2.3-1 shows, that in the momentum equations, the source terms comprise a "within-fluid" pressure-gradient term, a body force term, and two interfluid terms resulting from interactions between the two fluids.

Term F_v or F_w is the mean friction force per unit volume that fluid 1 exerts upon fluid 2 in the radial or axial direction. The other interfluid source term is associated with the entrainment m so that it represents the average entrainment of momentum through the interface of the two fluids.

For steady flows, the time-dependent terms in the equations disappear.

2.4 Modelling assumptions

In order to close the above set of equations, assumptions are required for μ_1 , m , F_v , F_w , D , B_y and B_z . The constant-effective-viscosity model and some assumptions of the two-fluid model are used to model them; these are described in the following subsections.

2.41 Constant-effective-viscosity model

The constant-effective-viscosity model suggested by Schlichting (1966) and also by Spalding (1979) is employed in the present study.

In this model, μ_1 is replaced by an effective viscosity μ_{eff} and for a round jet, μ_{eff} is given as:

$$\mu_{eff} = C_{\mu} \rho_1 W_i D_n \quad (2.41-1)$$

where C_{μ} is a constant, ρ_1 is the density of the fluid, W_i is the nozzle exit velocity; and D_n is the diameter of the nozzle exit.

The suitability of this model to the problems considered in the present study will be discussed.

2.42 Some assumptions of the two-fluid model

In addition to the constant-effective-viscosity model, some assumptions of the two-fluid model of turbulence [Spalding, 1982a] are used to model the intermittency phenomenon of the flows considered.

The detailed general theory of the two-fluid model and its various versions can be found elsewhere [Spalding, 1982a, 1984a, 1985a]. In the following subsections, the assumptions of the model which are incorporated into this mathematical model will be described.

(1) Interfluid mass transfer, m

The interfluid mass transfer rate per unit volume, m , is expressed as:

$$m = C_m \rho_2 r_1 r_2 |W_1 - W_2| / L \quad (2.42-1)$$

where C_m is an empirical constant; L is a length scale, characterising the mean fragment size.

(2) Mean fragment size, L

In the present study, it is assumed that the mean fragment

size L is equal to the "mixing length" , L_m , for miscible flows. At this stage, the simplification adopted for L appears reasonable, since L_m is representative of the large-scale turbulent eddies and possesses the necessary requirement that the entrainment process must lead to an increase in L . However, this aspect of the modelling is subject to future improvement, since it is preferable that L be a local quantity as discussed later.

The mixing length L_m for a turbulent round jet is given by Spalding (1979) as:

$$L_m = 0.1875 \times Y_{0.5} \quad (2.42-2)$$

where $Y_{0.5}$ is the half-radius of the round jet at which the streamwise velocity has one half its centreline velocity. For a turbulent round jet, $Y_{0.5}$ is given as:

$$Y_{0.5} = 0.0845 \times z \quad (2.42-3)$$

where z is the axial distance from the nozzle exit. Combining (2.42-2) and (2.42-3) gives the mixing length expression at each axial station of the jet as:

$$L_m = 0.01584 \times z \quad (2.42-4)$$

(3) Interfluid friction forces, F_v and F_w

The interfluid friction forces in the radial and axial directions , F_v and F_w, are expressed as:

$$F_v = f (V_1 - V_2) \quad (2.42-5)$$

$$F_w = f (W_1 - W_2) \quad (2.42-6)$$

where the "friction factor" , f, is given as:

$$f = C_f \rho_2 r_1 r_2 | W_1 - W_2 | / L \quad (2.42-7)$$

(4) Gradient-diffusion coefficient, D

The gradient-diffusion coefficient, D, represents the rate of instantaneous departure from the mean velocity due to vorticity fluctuations.

D is given as:

$$D = C_d \times \frac{\mu_{eff}}{\rho_1} \quad (2.42-8)$$

(5) The body force

The body force only affects fluid-1 in the axial direction. This body force, which is caused by the density difference between the two fluids, appears in the source terms as:

$$B_{1z} = g (\rho_1 - \rho_2) \quad (2.42-9)$$

where g is the acceleration due to gravity.

2.5 Empirical constants

Four empirical constants appeared in the modelling assumptions described above; they are C_μ , C_m , C_f , C_d . These constants may be estimated from the experimental data. Their final values should be determined by comparison of the predictions with the experimental data.

In this work, the values suggested by Spalding (1979, 1982a) and Malin (1983a) are used as "standard value". They are:

$$C_\mu = 0.0142, \quad C_m = 0.35, \quad C_f = 0.05, \quad C_d = 1.0.$$

2.6 Boundary and initial conditions

A closed system of equations is presented above; but the mathematical formulation is complete only when the relevant boundary and initial conditions are specified.

2.61 Boundary conditions

For the flows considered in the present study, the centreline of the flow is an axis of symmetry; consequently it is a zero-flux boundary.

At the outer boundary of the calculation domain, a fixed-pressure condition is applied. The two velocity components of both fluids are set to zero, to simulate the stagnant surrounding.

The boundary conditions at the inlet and outlet planes of the calculation domain are different for the various flow situations considered in this work. They will therefore, be given in the relevant sections later.

2.62 Initial conditions

For a transient calculation, the initial conditions

represent the physical conditions existing at the start of the transient.

Again, the initial conditions corresponding to each of the problems under consideration will be described in the relevant sections.

2.7 Closure

In this chapter, the partial-differential equations which govern the flows considered have been presented. The modelling assumptions made and the empirical constants used have been given. Also, the boundary and initial conditions have been outlined.

CHAPTER 3 - THE NUMERICAL SOLUTION PROCEDURE

3.1 Introduction

Chapter 2 presented a complete mathematical model. In cases of practical importance, however, it is impossible to obtain analytical solutions from these non-linear and strongly coupled equations. The task of this chapter is to describe a general numerical solution procedure which is employed for solving these equations.

The governing equations are numerically solved using the PHOENICS-84 computer code [Spalding, 1984b; Rosten and Spalding, 1984]. This embodies an implicit finite-domain formulation of the equations and a procedure for solving these equations which are the result of the work of Patankar and Spalding (1972).

The PHOENICS computer code is described in detail by Spalding (1982b). Here, only a brief description will be given.

The major steps of the solution procedure can be summarized as follows:

- (1) The flow domain is divided into many discrete regions by constructing a finite-difference grid which generates "control volumes", "cells" or "sub-

domains" for each variable.

- (2) For each variable at each cell, finite-domain equations are derived by integrating the partial-differential equations over the control volumes. Appropriate assumptions are made for the variation of the variables between grid nodes.
- (3) Iteration, as distinct from direct matrix inversion, is essential because the finite-domain equations are non-linear and coupled to each other.
- (4) The accuracy of the numerical solution for the finite-domain equations depends on the number of grid-nodes, and the time-steps employed in a transient problem. A "grid-independent" and "time-independent" solution may be obtained provided the number of the grid-nodes and time-steps is sufficiently large. Increasing the number increases also the computational expense. It is therefore, important to select an optimum number of grid-nodes and time-steps for accuracy and economy.

The remainder of this chapter consists of five sections. Section 3.2 describes the finite-difference grid while Section 3.3 gives the general finite-domain equations. Incorporation of the boundary and initial conditions is

described in Section 3.4. In Section 3.5, the "IPSA" solution procedure is briefly presented and finally, the numerical solution procedure is summarized in Section 3.6.

3.2 The finite-difference grid

3.21 General concept

The finite-difference grid consists of a set of lines (grid lines), which intersect normal to each other, over the entire region of interest. Thus, the whole integration domain is filled by a set of non-overlapping control cells which form reference locations for specifying the flow parameters.

Fig. 3.21-1 shows a typical two-dimensional cartesian control cell.

The geometrical centre P is called "grid-node". The neighbour points are defined as N , S , E , and W (North, South, East, and West), and the lines connecting P and its neighbouring points cut the cell faces at n , s , e , and w . The values of the variables at P , N , S , E , and W are regarded as representing the values within the whole cell, while the values at n , s , e , and w are regarded as representing values over the whole faces of the cell.

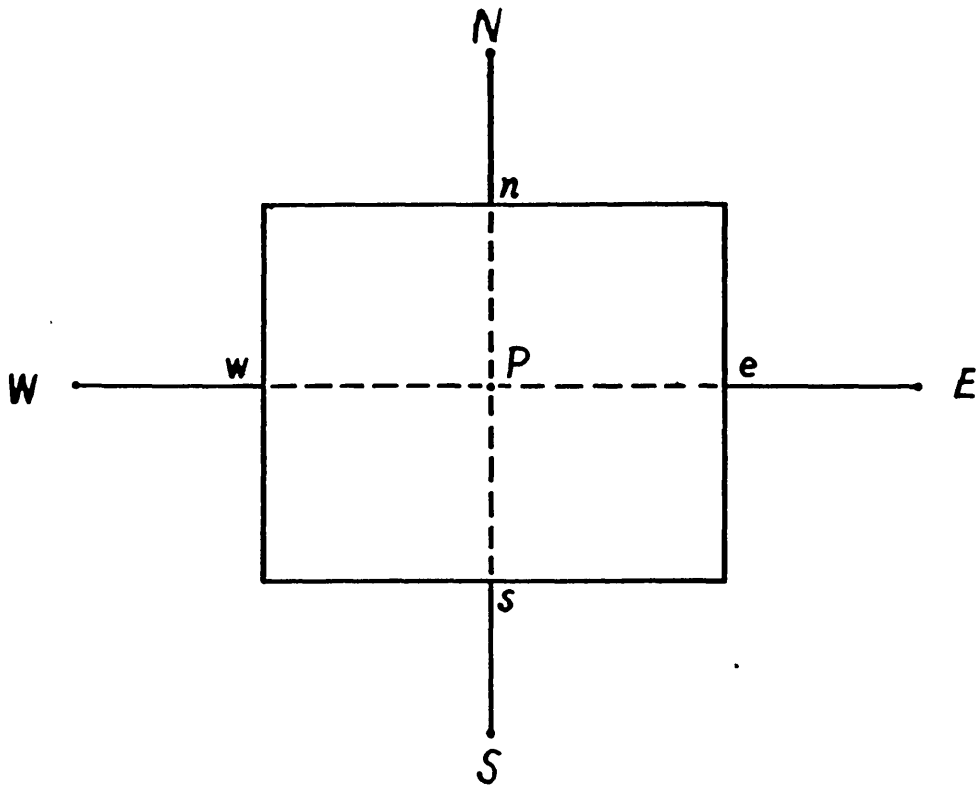


FIG. 3.21-1 A TWO-DIMENSIONAL CARTESIAN
CONTROL CELL

The grid lines may be distributed either uniformly or non-uniformly over the integration region. It is possible with a non-uniform distribution to locate more grid-nodes in regions where large gradients of the flow properties are expected; a more accurate solution may then be obtained without sacrificing economy.

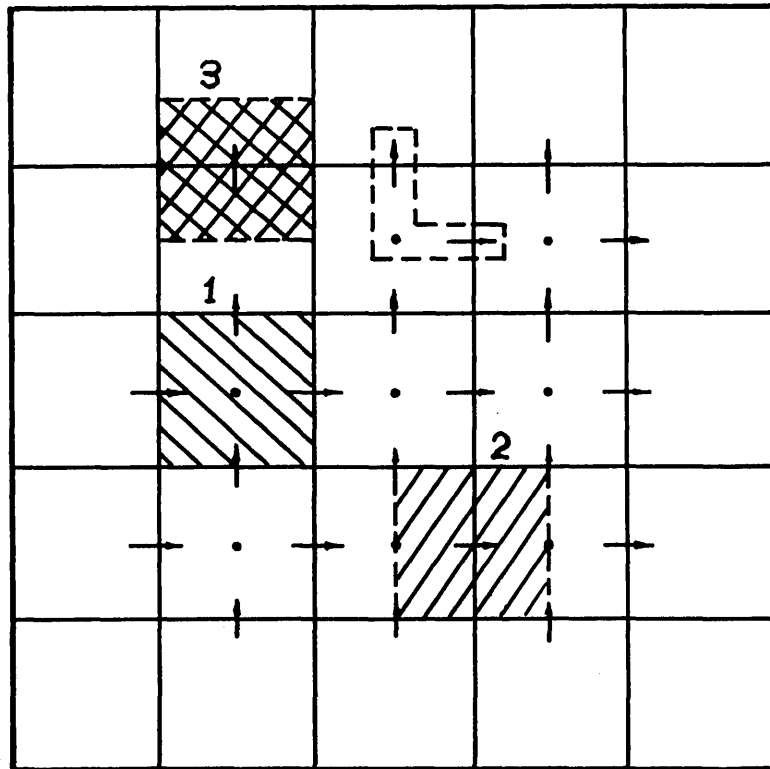
3.22 Location of flow variables

Fig. 3.22-1 shows the grid configuration and the locations at which the flow variables are stored.

With the exception of the velocities, the pressure and any scalar variables, such as temperature, are stored at the grid-nodes. The velocity components are calculated and stored at midway positions between the nodes. The locations of the velocity components are indicated by the arrows in Fig. 3.22-1.

This arrangement is known as a staggered-grid system (Harlow and Welch, 1965) . It has the following advantages:

- (1) The velocity components are conveniently located for the calculation of the convective fluxes of flow properties stored at the grid-nodes.
- (2) The calculation of a mass balance over the region surrounding a grid-node is made easy because the velocities normal to the boundaries of this region are located on the boundaries.
- (3) The pressures are located such that the calculation of pressure gradients that affect the velocities is easily made.



SYMBOL	LOCATION	CONTROL VOLUME FOR:
1		P, r
2	→	U
3	↑	V

FIG. 3.22-1 GRID SHOWING LOCATION OF VARIABLES
AND THEIR CONTROL VOLUMES

As shown in Fig. 3.22-1, the velocities are enclosed in dashed enclosures taking an "L" - shape; the purpose of these enclosures is to help in identifying the velocity components which refer to a particular grid-node.

3.23 Control volume

The term "control volume" refers to those imaginary regions over which the partial-differential equations are integrated so as to convert them into their finite-domain form.

Because there are three different types of locations for the variables in the "staggered-grid", there are three types of control volumes: (a) main or grid-node control volumes for the variables stored at each grid-node; (b) u-velocity control volumes for the u-velocities storage locations; and (c) v-velocity control volumes for the v-velocities storage locations.

Examples of these control volumes are also shown in Fig. 3.22-1. The boundaries of the grid-node control volumes lie halfway between the grid-nodes; thus, the surfaces of these control volumes pass through the locations where the normal velocities are stored. The velocity control volumes are bounded in a similar way on the side perpendicular to the velocity component; in the direction of the velocity component, however, the boundaries of the

control volumes pass through the grid-nodes in front of and behind the velocity locations.

3.3 The general finite-domain equation

The finite-domain equations are obtained by integrating the respective differential equations over a small control volume (known as a cell, etc.) surrounding each grid-node.

The integration procedure is now explained with reference to a general differential equation for the transport of a general variable ϕ . This equation is written as:

$$\frac{\partial}{\partial t} (\rho r \phi) + \text{div} (\rho r \vec{u} \phi) = \text{div} (\Gamma_{\phi} r \text{grad} \phi) + S_{\phi} \quad (3.3-1)$$

The nomenclature used in this section is the same as in Chapter 2, except that no subscripts are used to distinguish the variables of the two fluids.

In (3.3-1), ρ is the density, r is the volume fraction of the fluid considered, $\vec{u}$ is the velocity vector, ϕ is the variable being transported, Γ_{ϕ} is the exchange coefficient and S_{ϕ} is the source of ϕ .

The general finite-domain equation for the transport of a general variable is now derived by integrating equation (3.3-1) over the finite control volumes as follows:

$$\iiint_V \frac{\partial}{\partial t} (\rho r \phi) dv + \iint_A (\rho r \vec{u} \phi) \cdot d\vec{A} = \iint_A \vec{J}_\phi \cdot d\vec{A} + \iiint_V S_\phi dv \quad (3.3-2)$$

where V denotes the cell volume, A its surface area, and $\vec{J}_\phi$ the diffusion flux defined as:

$$\vec{J}_\phi = -\Gamma_\phi r \text{ grad} \phi \quad (3.3-3)$$

The transport of a variable ϕ across the faces and throughout time yields transient, convection, and diffusion terms. Other terms resulting from the integration of the general differential equation over the control volume are called source terms. These terms are now briefly discussed one by one.

(1) Transient term

Because ϕ is assumed to be uniform throughout the control volume, the integration of the transient term yields:

$$\iiint_V \frac{\partial}{\partial t} (\rho r \phi) dv = V \frac{\partial}{\partial t} (\rho r \phi) = \frac{V \rho}{\Delta t} (r \phi_p - r_o \phi_{p,o}) \quad (3.3-4)$$

where the subscript o refers to the value prevailing at the start of the time-step.

(2) Convection term

Since all the variables are presumed to be constant over each cell face, the surface integral of the convection term in equation (3.3-2) gives:

$$\iint_A (\rho r u \phi) \cdot d\vec{A} = \sum_{n,s,e,w} (\rho r u \phi) \cdot \vec{A} \quad (3.3-5)$$

Interpolations are required for the cell-face value of ϕ which is used in the surface integral (3.3-5), and the "upwind-differencing scheme" of Spalding (1972) is used to evaluate the convection fluxes across the control volume faces.

Equation (3.3-5) can be written in terms of the convection flux C as:

$$C_i \phi = \rho u_i A_i \phi \quad (3.3-6)$$

(3) Diffusion term

Again, all the variables are presumed to be constant over each cell face so that the surface integral of the diffusion term in equation (3.3-2) gives:

$$\oint_A \vec{J}_\phi \cdot d\vec{A} = \sum_{n,s,e,w} \vec{J}_\phi \cdot \vec{A} \quad (3.3-7)$$

With the aid of the general flux law, i.e. equation (3.3-3), this term may be expressed as:

$$\sum_{n,s,e,w} \vec{J}_\phi \cdot \vec{A} = - \sum_{n,s,e,w} \Gamma_\phi r \text{grad}_\phi \cdot \vec{A} \quad (3.3-8)$$

or expressed in terms of the diffusion flux D as:

$$D_i \phi = A_i \Gamma_i r \phi / (X_i - X_p) \quad (3.3-9)$$

where $(X_i - X_p)$ is the distance from the in-cell grid-node P to the relevant neighbour node.

(4) Source term

The source terms of ϕ are linearised so that they may be written in the form:

$$S_{\phi} V = S_u V + S_p V \phi_p \quad (3.3-10)$$

The equation resulting from the sum of the various terms which have appeared in the above derivations can be written in the following general finite-domain equation:

$$A_p \phi_p = A_n \phi_n + A_s \phi_s + A_w \phi_w + A_e \phi_e + A_o \phi_o + S_u \quad (3.3-11)$$

$$A_p = A_n + A_s + A_w + A_e + A_o + (-S_p) \quad (3.3-12)$$

where A's denote the "coefficients" of the finite-domain equation; they are functions of the C's and D's in equation (3.3-6) and (3.3-9).

The subscripts have the following meanings:

- P Typical point (i.e. node) within cell;
- N North neighbour node;
- S South neighbour node;

W West neighbour node;
E East neighbour node;
O Grid node at earlier time.

Further details about the derivations can be found in "Four lectures on the PHOENICS computer code" [Spalding, 1982b] .

3.4 Incorporation of boundary and initial conditions

To obtain a solution of the finite-domain equations, appropriate boundary conditions, and initial conditions for a transient problem, have to be specified. The methods of implementing these conditions in the solution scheme are given below.

3.41 Boundary conditions

The values of the dependent variables at the boundaries of the integration domain must be specified throughout the solution procedure. These boundary conditions, which determine the uniqueness of the solution for a given flow situation, are usually of two types:

- (1) The value of the dependent variables itself is specified at the boundary. This is known to be the most stable of boundary conditions; and

- (2) The value of ϕ at the boundary is deduced from the known gradient at the boundary.

In the present study, both boundary conditions are handled by way of linearised sources. The details of this approach may be found in Spalding (1982b) so that only a brief description is given here.

The PHOENICS expresses the boundary conditions in a linear form, so that the total source is expressed as:

$$S = \dot{m}\phi_{BC} + C (\phi_{BS} - \phi_p) \quad (3.41-1)$$

where $\dot{m}$ is the prescribed inflow rate of mass to the cell;

ϕ_{BC} is the prescribed quantity having the dimensions of ϕ ;

ϕ_p is the unknown value of ϕ for the cell; and

c is a prescribed coefficient, always positive.

3.42 Initial conditions

For a transient problem, ϕ_0 in the general finite-domain equation (3.3-11) at the first time step represents the values of the dependent variables at the start. The values of the variables at the beginning of the transient, therefore, must be specified over the whole domain to initiate the calculation.

Thus, all the dependent variables are given the physical conditions existing at the start of the transient.

3.5 The "IPSA" solution procedure

"IPSA" stands for "interphase slip algorithm". It is a solution procedure which handles the presence of two simultaneously-present phases, which share a pressure. "IPSA" is employed in PHOENICS as the apparatus to solve the mathematical formulations with the two-fluid assumptions.

A detailed description of "IPSA" and its PHOENICS version is provided by Spalding (1982b). Here the whole hydrodynamic solution cycle is briefly outlined as follows:

- (1) Solve for r_1 and r_2 from the two continuity equations.
- (2) With guessed pressures, solve for the velocities from the momentum equations.
- (3) Insert r 's and velocities into the joint continuity equation, so as to establish a representative continuity error for each cell.
- (4) Differentiate the joint continuity equation with

respect to pressure so as to derive the linear equation for pressure corrections:

$$L \{ p'_p, p'_n, p'_s, p'_e, p'_w, p'_h, p'_l \} = 0 \quad (3.5-1)$$

- (5) Solve in the same manner as employed in SIMPLEST. SIMPLEST stands for SIMPLE-shortened where SIMPLE stands for "semi-implicit method for pressure-linked equation" [Spalding, 1982b].
- (6) Apply corresponding corrections to pressures and velocities.
- (7) Repeat until the errors have sufficiently diminished.

3.6 Summary of the numerical solution procedure

The construction of the finite-difference grid, the derivation of the finite-domain form of the partial-differential equations and their method of solution have been presented in this chapter. The features of the solution procedure are summarized as follows:

- (1) - provision of an orthogonal finite-difference grid;
- (2) - use of a non-uniformly spaced grid;
- (3) - use of staggered storage locations;

- (4) - derivation of the finite-domain equations by integrating the differential equations over finite control volumes;
- (5) - formulation of the finite-domain equations in a general form;
- (6) - use of the IPSA solution procedure; and
- (7) - solution of the finite-domain equations by means of the PHOENICS computer code.

With this chapter, the theoretical formulation of the present study has been completed.

CHAPTER 4 - COMPUTATION OF AN AIR-INTO-AIR STEADY JET

4.1 Introduction

The mathematical model was presented in Chapter 2, while the numerical solution procedure was given in Chapter 3. The attention is now turned to investigating the flows considered in this thesis one by one.

Although the main subject of the present study concerns transient jets, the first flow considered, however, is a steady jet. The objectives of this study are:

- (1) To test and validate the mathematical model and its numerical solution procedure incorporated in the PHOENICS-84 computer code by comparing the computational results with available experimental data for a steady jet; and
- (2) To obtain predictions of a steady jet for comparison with those of a transient jet which will be considered in the next chapter.

In this chapter, a well-known turbulent flow will be focussed upon, namely, a turbulent, round, air-into-air steady jet resulting from the steady injection of air through a vertically-directed nozzle into an atmosphere at rest. Because the injected fluid (air) has the same

density as the atmosphere, the resulting jet forms a conical region, extending to infinity, in which the momentum flux across any horizontal plane is a constant.

This flow has been studied both experimentally and theoretically, and in many different ways. Conventional turbulence models still have difficulty with it; for agreement between predictions and experiments can be achieved only by way of an ad hoc adjustment to the "constants" of the model. In addition, these models can not provide a prediction for the intermittency .

Computations of this flow have been carried out for various nozzle exit conditions (nozzle diameter and nozzle exit velocity). Here the quantities of interest are the mean velocity along the centre-line, the profiles of the streamwise velocities and intermittency factor, and the rate of spread in the self-similar region of the flow.

The mean flow of this jet was experimentally studied by numerous authors. These were extensively reviewed by Rodi (1972) . In the present study, the data of the mean flow of Rodi (1972) and Wygnanski and Fiedler (1969) are taken to be compared.

In the laboratory, "conditional-sampling" techniques have been developed which discriminate between the

turbulent and non-turbulent zones of a turbulent shear flow. With those techniques, the experimentalist is able to perform measurements of flow variables by averaging separately over the turbulent and non-turbulent parts of the flow. These measurements supplement the usual unconditioned measurements obtained by conventional time averaging.

Following Malin's review (1983a), the data of Chevray and Tutu (1978) for the similarity-profiles of the streamwise velocities of the turbulent and non-turbulent fluids, and the data of Wygnanski and Fiedler for the similarity-profile of intermittency are to be compared.

The remainder of this chapter consists of six sections, in which Section 4.2 describes the flow conditions, while Section 4.3 gives all the computational details. Section 4.4 presents the main results and comparisons with experimental data. Section 4.5 shows the effect of the empirical constants of the two-fluid assumptions on the predictions, followed by the discussion and conclusion in Sections 4.6 and 4.7 respectively.

4.2 The flow considered

A turbulent, round, air-into-air, steady jet is considered. The flow situation is sketched in Fig. 4.2-1, and can also be described by the following:-

- air is injected;
- the surrounding air is at rest;
- the flow is steady;
- buoyancy is absent;
- axial symmetry prevails; and
- the flow is turbulent.

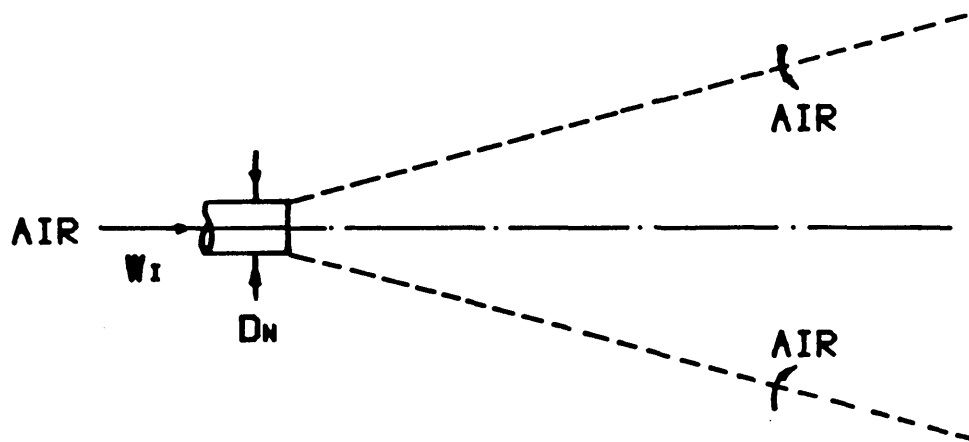


FIG. 4.2-1 A ROUND AIR-INTO-AIR JET

The flows with the following parameters are to be predicted:

- nozzle diameter D_n : 8 mm; 12.7 mm; 25.4 mm;
- nozzle exit velocity W_i : 30.48 m/sec.; 60.96 m/sec.; 121.92 m/sec.; 500 m/sec.; and
- the range of the Reynolds number Re : from 77,000 to 400,000.

The Reynolds number is defined as:

$$Re = \frac{W_i D_n}{\nu} \quad (4.2-1)$$

where: W_i is the nozzle exit velocity;

D_n is the diameter of the jet nozzle; and

ν is the kinematic molecular viscosity of air.

4.3 Computational details

4.31 Solution procedure and calculation domain

Chapter 2 outlined a closed system of equations. These equations describe the flows considered in this thesis. For the air-into-air steady jet to be predicted in this chapter, however, a further assumption has been made in order to permit an economic usage of both computer storage and computer time. This assumption is that the viscous transport of the momentum in the axial direction is negligible. The flow can now be treated as a "parabolic flow" .

The flow is predicted by the forward-marching, iterative, finite-difference solution procedure embodied in the PHOENICS-84 computer code. This procedure involves the use of the two-phase-flow algorithm embodied within the program, and the adopted convention is that fluid-1 refers

to the turbulent (rotational) fluid and fluid-2 refers to the non-turbulent (irrotational) fluid.

Fig. 4.31-1 shows the integration domain for the calculation. The flow is symmetrical about the flow axis so that the calculation needs only be performed over one-half of the flow. The marching integration procedure starts from prescribed values of all dependent variables at the inlet of the calculation domain. This inlet is located at the discharge-plane of the jet nozzle, where the outer boundary of the domain is located in the free stream 1.25 radius of the nozzle away from the flow axis.

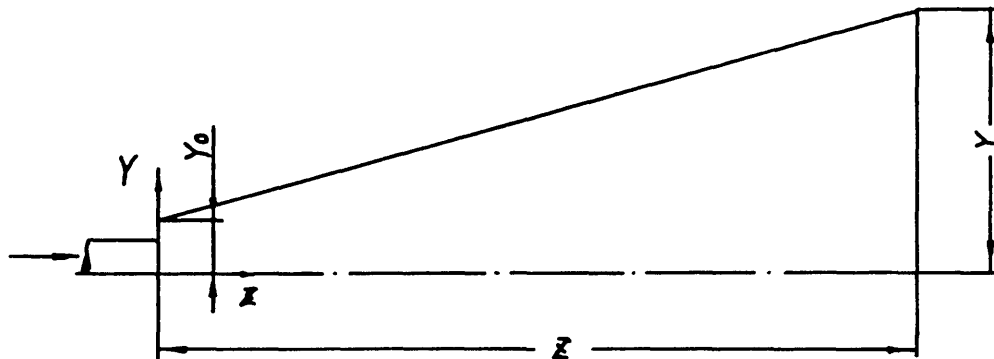


FIG. 4.31-1 THE CALCULATION DOMAIN FOR THE
AIR-INTO-AIR STEADY JET

The width of the outer boundary of the domain changes with axial distance; therefore, the domain covers only that region where the properties of the jet vary.

In the present calculation, the outer boundary width Y expands with the axial distance Z according to:

$$Y = Y_0 + zf \quad (4.31-1)$$

where Y_0 is the value of Y at the inlet plane and f , the spreading rate of the outer boundary, is taken as 0.225 and is deduced from the half-radius spread rate of the round jet presented by Rodi (1972). It should, once again, be emphasized that f here prescribes the spread of the calculation domain and not the jet itself, the latter being an outcome of the calculation.

Thus, the edge of the calculation domain always lies outside the edge of the flow. To achieve an appropriate spread of the calculation domain, the domain-expansion formulae mirror those for the length scale of the flow.

The downstream-marching integration process is carried out until self-similarity is attained.

4.32 The Grid

Initially, calculations were performed to assess: (a) the effect of radial grid-size; and (b) the effect of forward step-size on the prediction.

Results are presented to show the grid-size effects on: (a) the mean velocity along the centre-line; (b) the similarity-profiles of streamwise velocities including those of the turbulent fluid, non-turbulent fluid and mean flow; and (c) the similarity-profile of the intermittency factor.

In testing the radial grid-size effect, uniformly distributed grid-nodes with $NY=30, 35, 40, 50$ have been used. These calculations were performed with a forward step-size $DZ/Y=0.1$, where DZ is the axial step-size and Y is the width of the outer boundary. Fig. 4.32-1 shows the effect of successive radial grid-size refinements on the mean velocity along the centre-line. Figs. 4.32-2 and 4.32-3 show respectively the effect of radial grid-size on the similarity-profiles of the streamwise velocities and intermittency factor.

From these figures, it can be seen that all the quantities tested for are insensitive to the range of radial grid-sizes considered, and that using 35 radial grid-nodes provides sufficient accuracy.

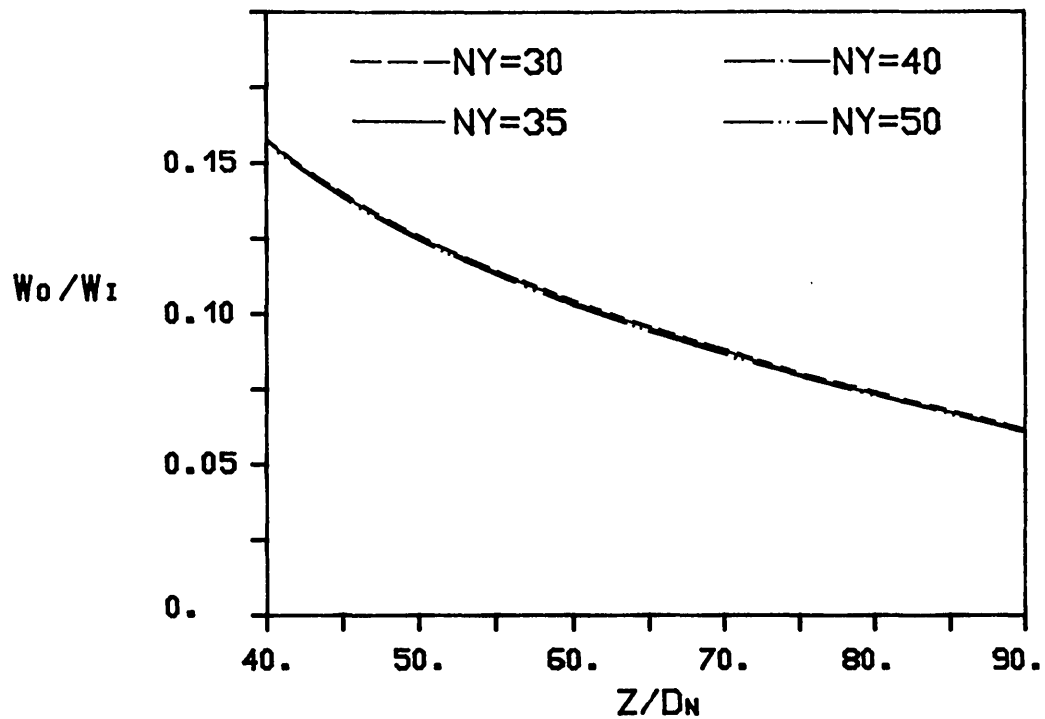


FIG. 4.32-1 EFFECT OF RADIAL GRID-SIZE ON THE MEAN VELOCITY ALONG THE CENTRE-LINE

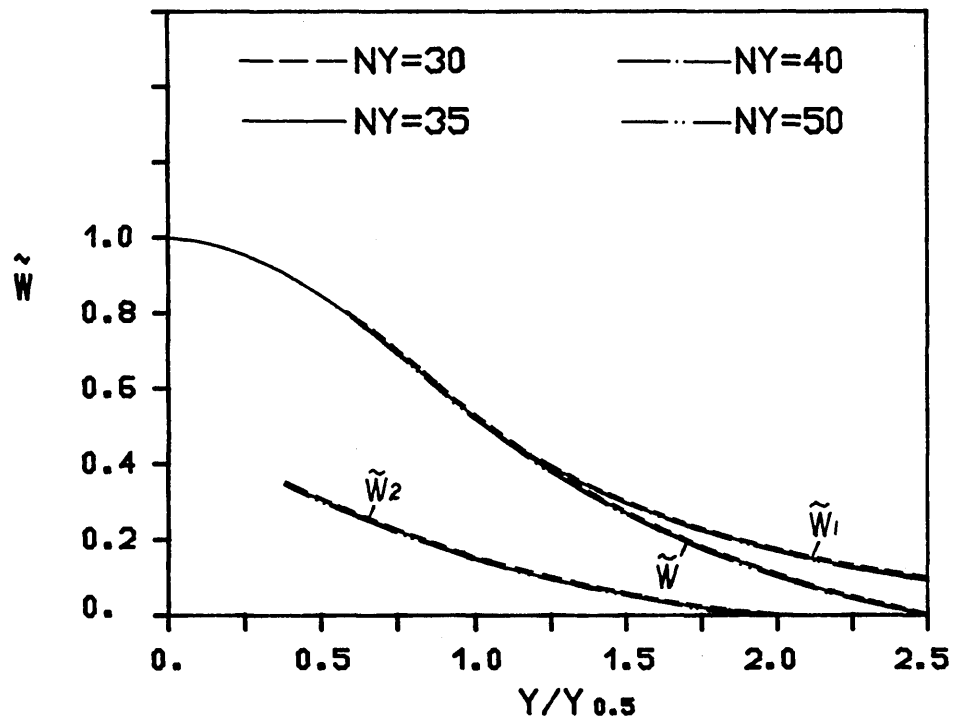


FIG. 4.32-2 EFFECT OF RADIAL GRID-SIZE ON THE PREDICTED SIMILARITY-PROFILES OF THE STREAMWISE VELOCITIES

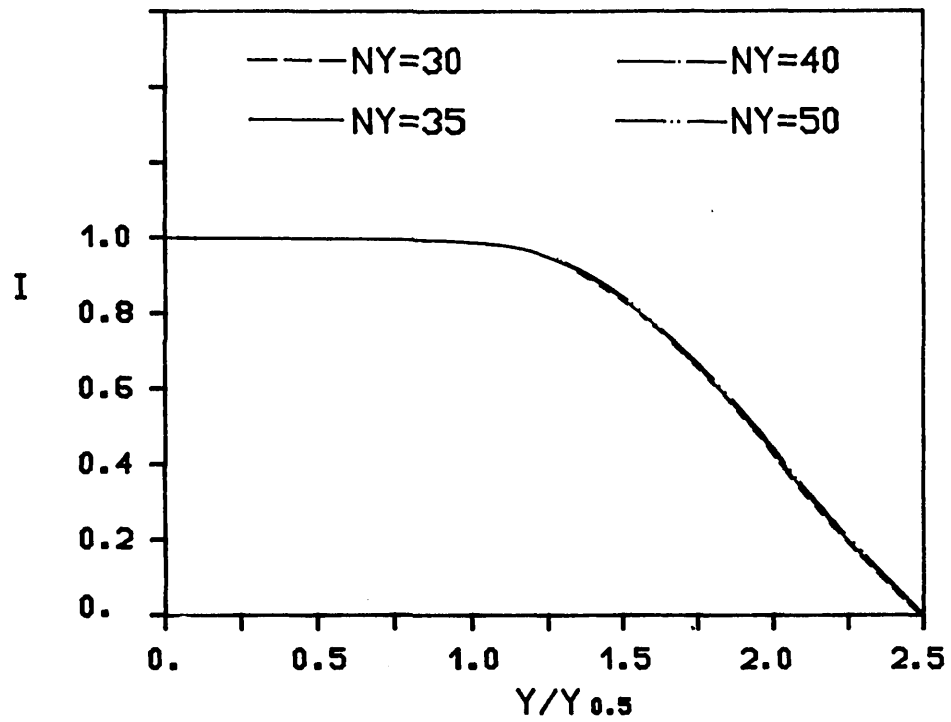


FIG. 4.32-3 EFFECT OF RADIAL GRID-SIZE ON THE PREDICTED SIMILARITY-PROFILE OF THE INTERMITTENCY FACTOR

In the calculations, the ratio of the forward step-size to the width of the outer boundary was set to a constant so that the forward step-size expanded proportionally with the width of the outer boundary. $DZ/Y=0.2$, 0.1 , 0.05 have been tested. Fig. 4.32-4 shows the effect of the forward step-size on the mean velocity along the centre-line. Figs. 4.32-5 and 4.32-6 show respectively the effect of the forward step-size on the similarity-profiles of the streamwise velocities and intermittency factor.

It can be seen from these figures that the mean velocity along the centre-line and the similarity-profiles of the streamwise velocity of the turbulent fluid and intermittency are insensitive to the forward step-sizes tested, while $DZ / Y = 0.1$ is required for sufficient accuracy of the similarity-profile of the streamwise velocity of the non-turbulent fluid.

In addition, no discernible change in the predicted spread rate was observed in all the calculations mentioned above.

In the following calculations, all the main results were obtained by using 35 grid-nodes in the radial direction and a forward step-size $DZ / Y = 0.1$ in the axial direction.

4.33 Boundary conditions

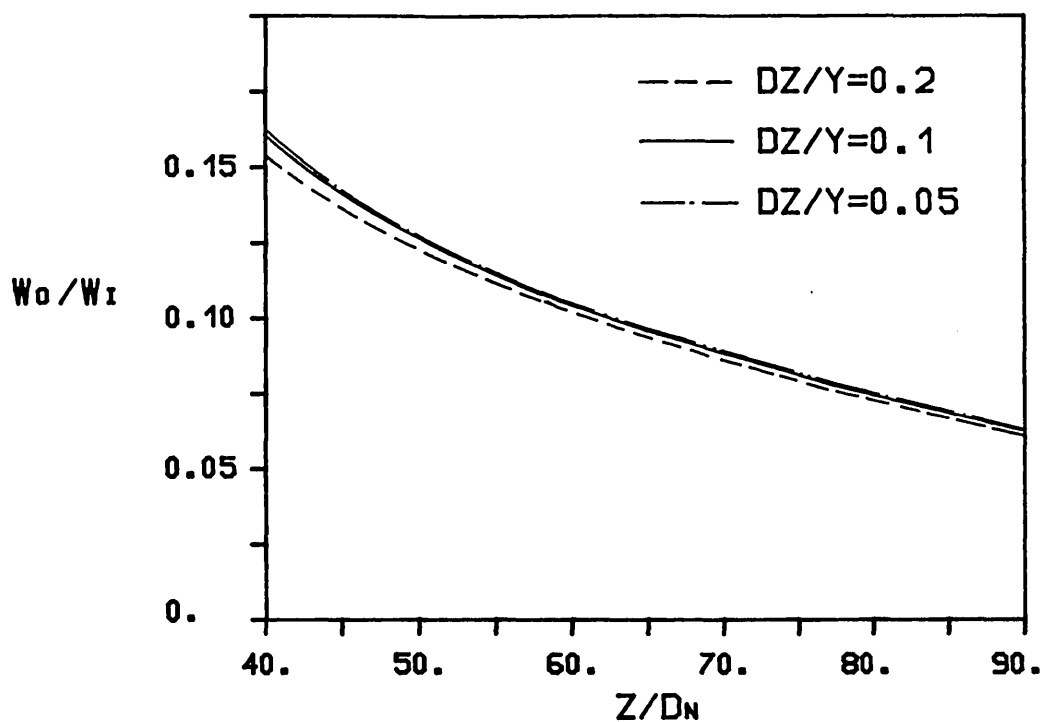


FIG. 4.32-4 EFFECT OF FORWARD STEP-SIZE ON THE MEAN VELOCITY ALONG THE CENTRE-LINE

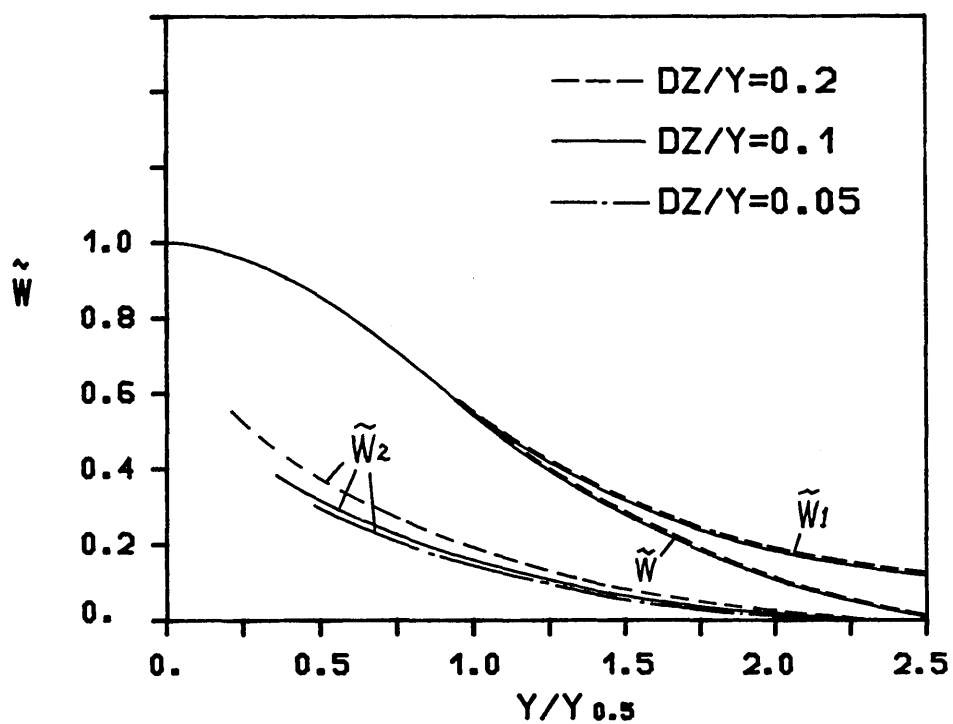


FIG. 4.32-5 EFFECT OF FORWARD STEP-SIZE ON THE PREDICTED SIMILARITY-PROFILES OF THE STREAMWISE VELOCITIES

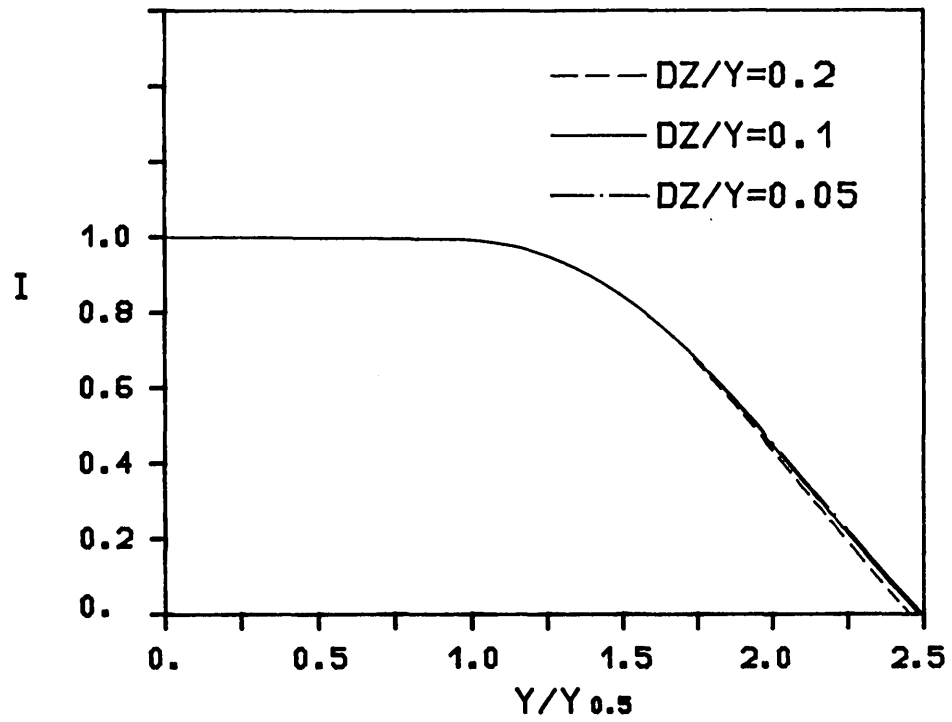


FIG. 4.32-6 EFFECT OF FORWARD STEP-SIZE ON THE
PREDICTED SIMILARITY-PROFILE OF
THE INTERMITTENCY FACTOR

Fig. 4.33-1 shows a sketch of the boundaries which are to be specified for the calculations in the present chapter.

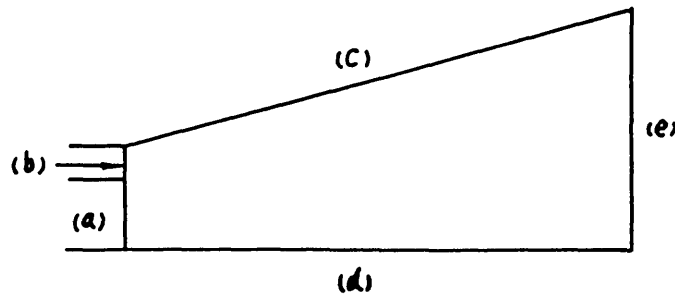


FIG. 4.33-1 THE BOUNDARIES OF THE CALCULATION DOMAIN FOR THE AIR-INTO-AIR STEADY JET

These boundary conditions are as follows:-

- Boundary (a) is the inlet of the jet. A fixed mass flow rate of the turbulent fluid is specified at this boundary. The description of this boundary will be given in detail in the next subsection.
- Boundary (b) is at the inlet plane, but outside the nozzle. A fixed-pressure boundary condition is applied.
- Boundary (c) is the outer boundary. As for boundary (b), a fixed-pressure boundary condition is specified. At the boundaries (b) and (c), the entrainment of surrounding

air across the boundaries is calculated from continuity. Velocity components in both directions of the two fluids are taken to be zero at these two boundaries, to simulate the stagnant surrounding.

- Boundary (d) is the inner boundary. Because the flow is axisymmetrical, a zero-flux boundary condition is specified.

- Boundary (e): no boundary condition is required for this boundary because the flow is considered to be a parabolic flow.

4.34 Inlet boundary conditions

The calculation starts at the discharge-plane where top-hat profiles are assumed for the streamwise velocity and intermittency factor. Within the jet nozzle, it is assumed that only turbulent fluid is present, i.e. the intermittency factor I is equal to 1.

At the inlet, it is also assumed that the streamwise velocity of the turbulent fluid is equal to that of the mean flow, and that radial velocity components are negligible.

It should be noted that arbitrary inlet profiles may be

chosen for the calculation because the similarity solution is independent of the prescribed inlet profiles. Of course, upstream of the similarity region, in the region of flow development, the flow field is influenced by the prescribed profiles of the dependent variables.

4.35 Similarity consideration

A test has also been made for the similarity behaviour of the predictions. Figs. 4.35-1 and 4.35-2 show respectively the predicted profiles of the streamwise velocities and intermittency at $Z / D_n = 46, 58, \text{ and } 73$. It can be found that the flow is already self-similar at about 60 nozzle diameters from the nozzle exit.

4.36 Convergence

The solution procedure requires slab iterations on all dependent variables at each forward step. The convergence criterion used was the level at which subsequent iterations were shown to cause negligible changes in the values of the monitored dependent variables. Checks were also made at various downstream locations on the overall mass imbalance within each fluid.

To secure convergence of the calculations the following practices were employed in the present computations:

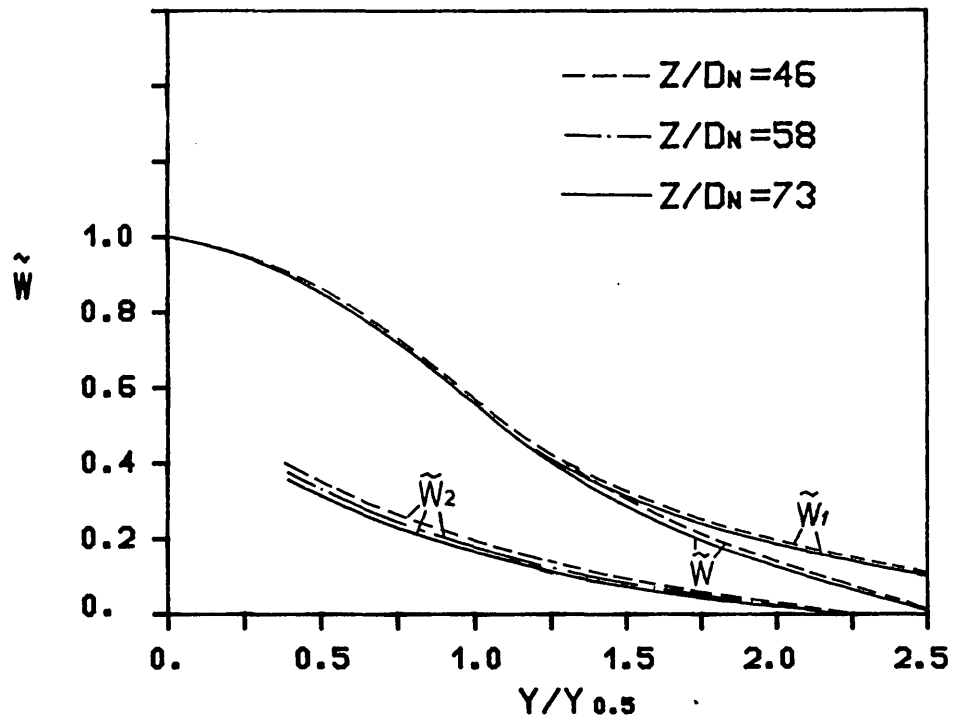


FIG. 4.35-1 TEST FOR SIMILARITY ON THE PREDICTED PROFILES OF THE STREAMWISE VELOCITIES

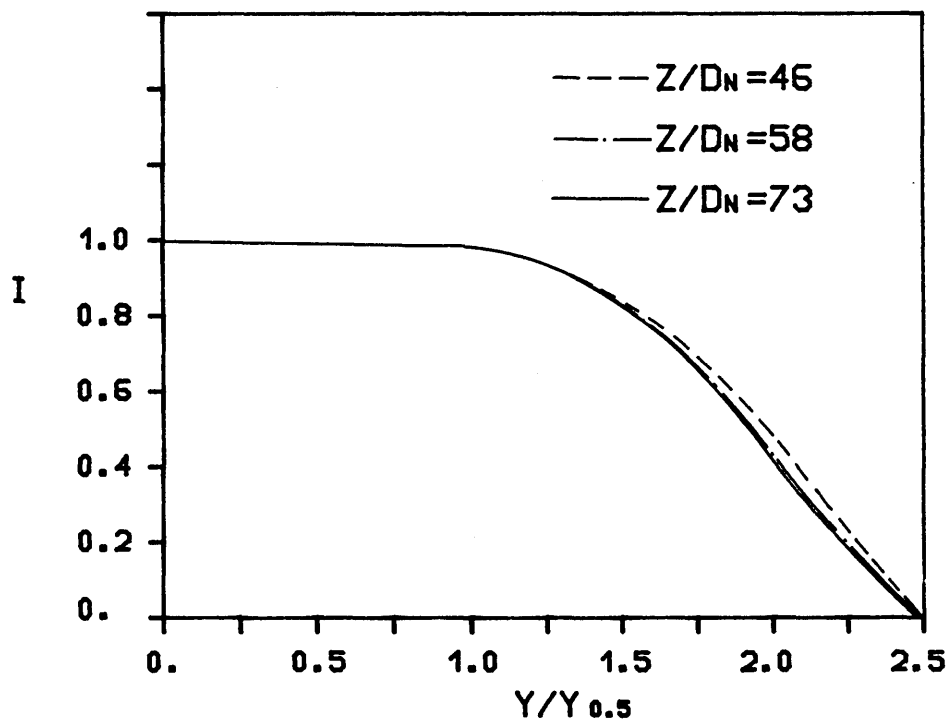


FIG. 4.35-2 TEST FOR SIMILARITY ON THE PREDICTED PROFILE OF THE INTERMITTENCY FACTOR

(1) It was assumed that there was no slip between the two fluids at the first 25 forward steps where the calculation reached an axial distance of about 2.5 nozzle diameters. One of the causes for the divergence of the calculation was the assumption of the "top-hat" profiles for the streamwise velocity and intermittency factor of the turbulent fluid at the inlet. To overcome this difficulty, it was necessary to prevent slip between the two fluids at the first few forward steps. This practice was satisfactory in the present calculations, since only the quantities in the self-similar region were of interest.

(2) Inertial relaxation on all dependent variables was required. Typically, a "false time-step" of 0.001 in the PHOENICS-84 computer code was employed to relax the streamwise velocity components.

(3) The change in the intermittency between successive iterations was limited. Typically, a limitation of 10% usually proved sufficient.

25 iterations were required at each slab, and at $I_Z = 180$, the sum of the absolute values of the flow residuals was of the order of 10^{-6} .

Table 4.36-1 summarizes all the information relating to convergence in this calculation.

Variable	Value
Grid	35 grid-nodes in Y-direction DZ/Y=0.1 in Z-direction
Type of grid spacing	uniform in Y-direction expanding in Z-direction
Iteration number at each slab	25
Relaxation	FALSDT: 0.001 for W_1, W_2 LINRLX: 0.1 for r_1, r_2
Slab-wise sum of ABS(vol. flow residuals)	3.191×10^{-6}

Table 4.36-1 Computational details for the
air-into-air steady jet

4.37 Computer storage and computer time

In this section, the computational requirements for the present calculations are presented.

Since the flow evolves parabolically in the streamwise direction computer storage was only required for the

previous and current axial slabs. This storage and the computer time required for a converged solution to arise, were direct functions of the numbers of the grid-nodes used and the variables to be solved for.

Table 4.37-1 displays the essential incore space and computer time for different numbers of grid-nodes in the radial direction and 180 forward steps.

Lateral grid-node number	Computer storage	Computer time(sec.)
30	3,509	1,509
35	3,939	1,718
40	4,369	1,907
50	5,229	2,281

Table 4.37-1 Computational requirement for
the air-into-air steady jet
(Computer used: Perkin Elmer
3220)

4.4 Main results

In this section, the computed values of the mean velocity along the centre-line, the similarity-profiles of the streamwise velocities and intermittency, and the rate of jet spread are presented and compared with experimental data.

(1) Mean velocity along centre-line

As shown in equation (2.22-1), the mean streamwise velocity is obtained from:

$$W = r_1 W_1 + r_2 W_2 \quad (4.4-1)$$

The mean velocity along the centre-line beyond 40 nozzle diameters is shown in Fig. 4.4-1. In this figure, W_0 is the mean velocity along the centre-line, W_i is the nozzle exit velocity, Z is the axial distance away from the nozzle exit, and D_n is the diameter of the nozzle. The measurements of Rodi (1972) are also shown in this figure.

(2) Similarity-profiles of streamwise velocities

Fig 4.4-2 shows the similarity-profiles of the streamwise velocities, which include those of the turbulent fluid, non-turbulent fluid, and the mean flow.

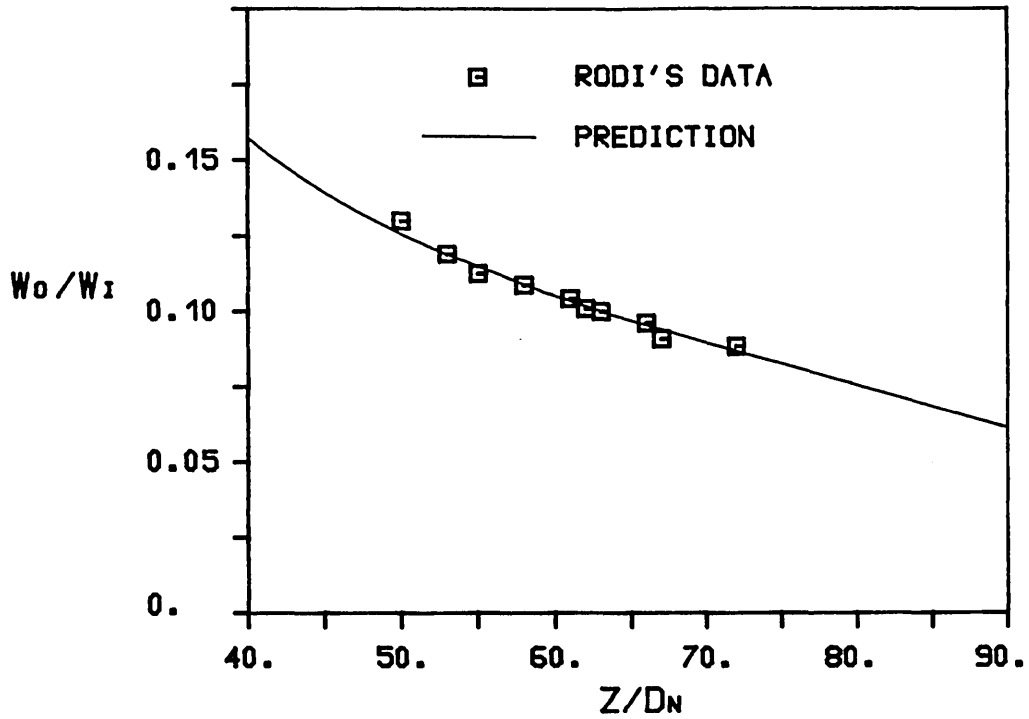


FIG. 4.4-1 MEAN VELOCITY ALONG THE CENTRE-LINE

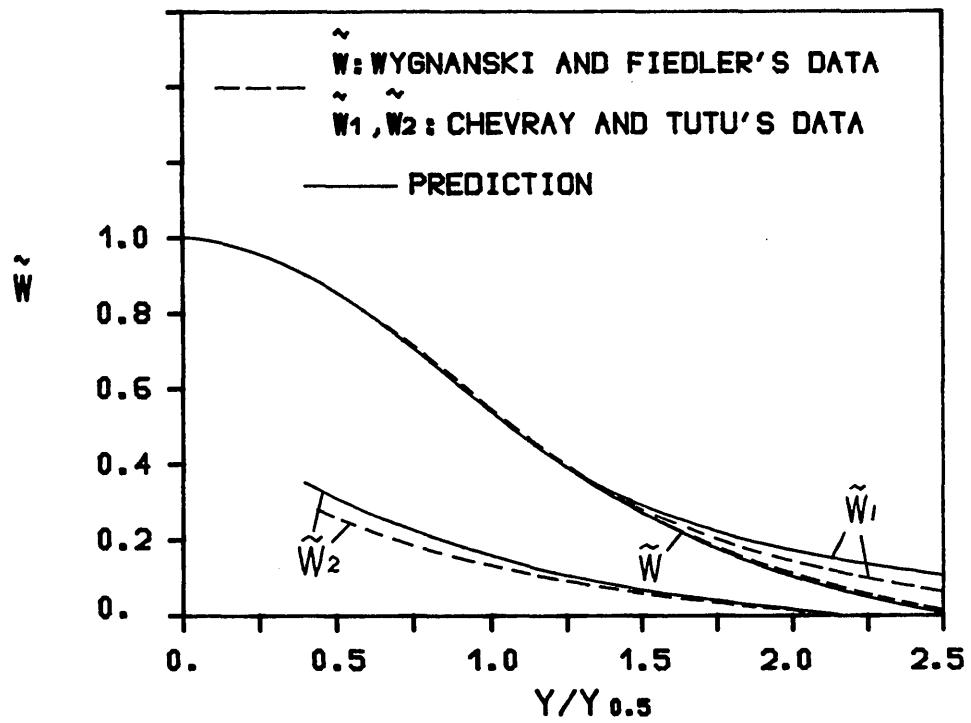


FIG. 4.4-2 SIMILARITY-PROFILES OF THE STREAMWISE-VELOCITIES

The dimensionless variables in this figure are defined as:

$$\tilde{W} = \frac{W}{W_0} \quad (4.4-2)$$

$$\tilde{W}_1 = \frac{W_1}{W_0} \quad (4.4-3)$$

$$\tilde{W}_2 = \frac{W_2}{W_0} \quad (4.4-4)$$

$Y_{0.5}$ is the half-radius of the jet at which the streamwise velocity of the mean flow has one half its axial value.

The data of Wygnanski and Fiedler (1969) for the similarity-profile of the streamwise velocity of the mean flow and the data of Chevray and Tutu (1978) for those of the turbulent and non-turbulent fluids are also shown in the figure for comparison.

(3) Similarity-profile of intermittency

As stated in Section 2.21, the computed volume fraction of the turbulent fluid r_1 , is interpreted as the intermittency I of the flow.

Fig. 4.4-3 gives the computational result and experimental data of Wygnanski and Fiedler (1969) of the similarity-profile of the intermittency.

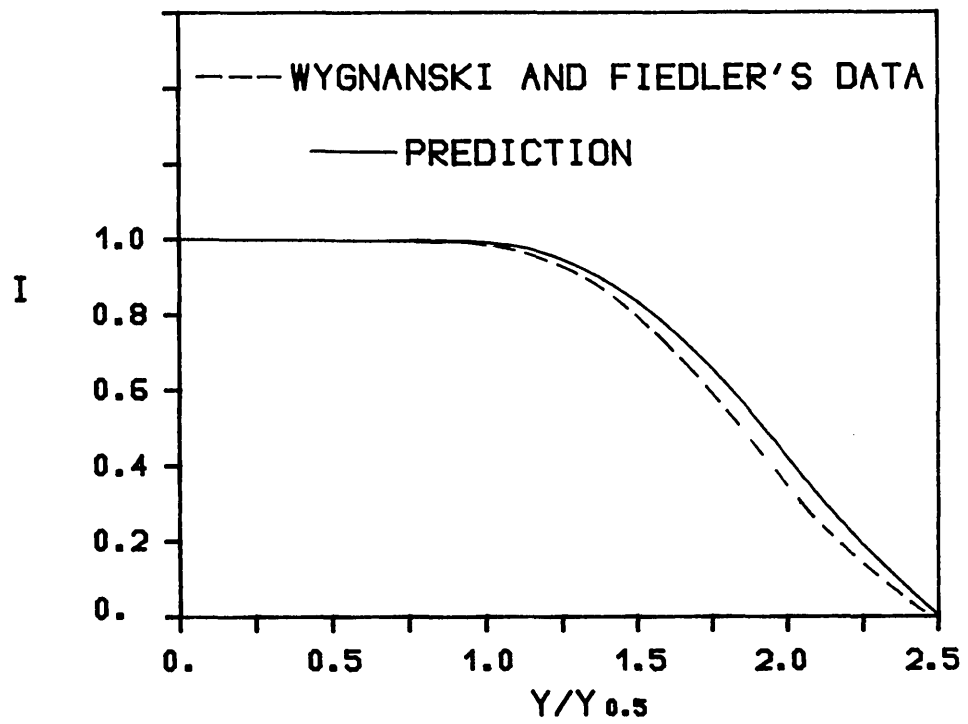


FIG. 4.4-3 SIMILARITY-PROFILE OF THE
INTERMITTENCY FACTOR

(4) Rate of spread

The rate of spread is expressed by $Y_{0.5} / Z$, where $Y_{0.5}$ is the half-radius of the jet and Z is the axial distance.

The values of the half-radius of the jet were calculated at each forward step during the course of the computation. In the similarity-region, the computed value of the rate of spread is 0.08, while that of the experimental data is 0.085 [Spalding, 1979]. Table 4.4-1 shows the predicted and measured values.

Rate of jet spread	Experimental data	Predicted value
$Y_{0.5}/Z$	0.085	0.08

Table 4.4-1 Spread rate of the air-into-air steady jet

4.5 Effect of empirical constants of two-fluid assumptions

Some assumptions of the two-fluid model have been

incorporated into the mathematical model. The effect of the empirical constants of these assumptions on the predictions are demonstrated in this section. The effect of varying these constants on the predicted rate of spread is summarized in Table 4.5-1. The changes in the rate of spread shown in Table 4.5-1 were brought about by changing the value of the constant as indicated with the remainder of the constants being their "standard values".

Rate of spread	Cm			Cf			Cd	
	0.2	0.35	0.5	0.02	0.05	0.1	0.7	1.0
$Y_{0.5}/Z$	0.08	0.08	0.08	0.083	0.08	0.079	0.08	0.08

Table 4.5-1 Effect of varying the empirical constants
on the spread rate of the air-into-air
steady jet

Figs. 4.5-1, 4.5-2, and 4.5-3 show respectively, the effects of Cm on the mean velocity along the centre-line, the predicted similarity-profiles of streamwise velocities, and the intermittency.

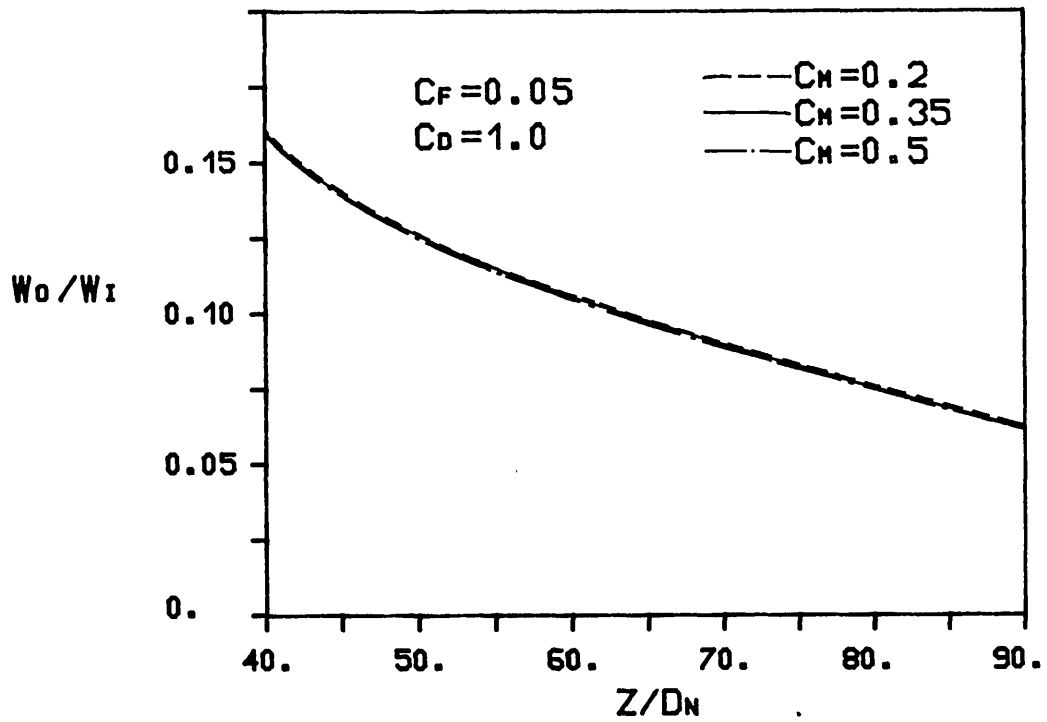


FIG. 4.5-1 EFFECT OF C_M ON THE MEAN VELOCITY ALONG THE CENTRE-LINE

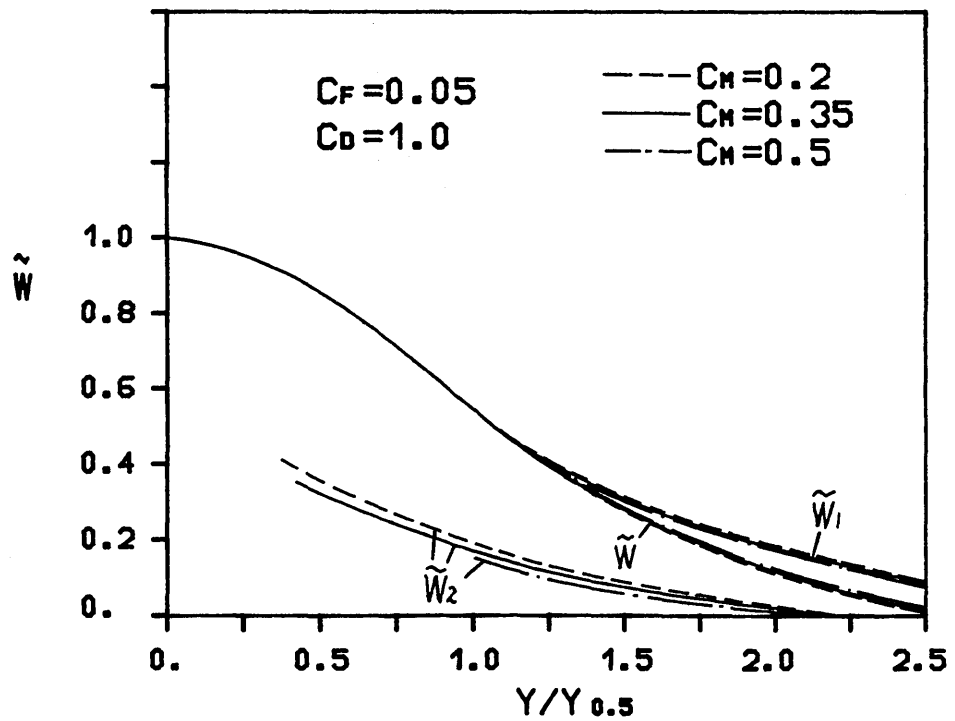


FIG. 4.5-2 EFFECT OF C_M ON THE PREDICTED SIMILARITY-PROFILES OF THE STREAMWISE VELOCITIES

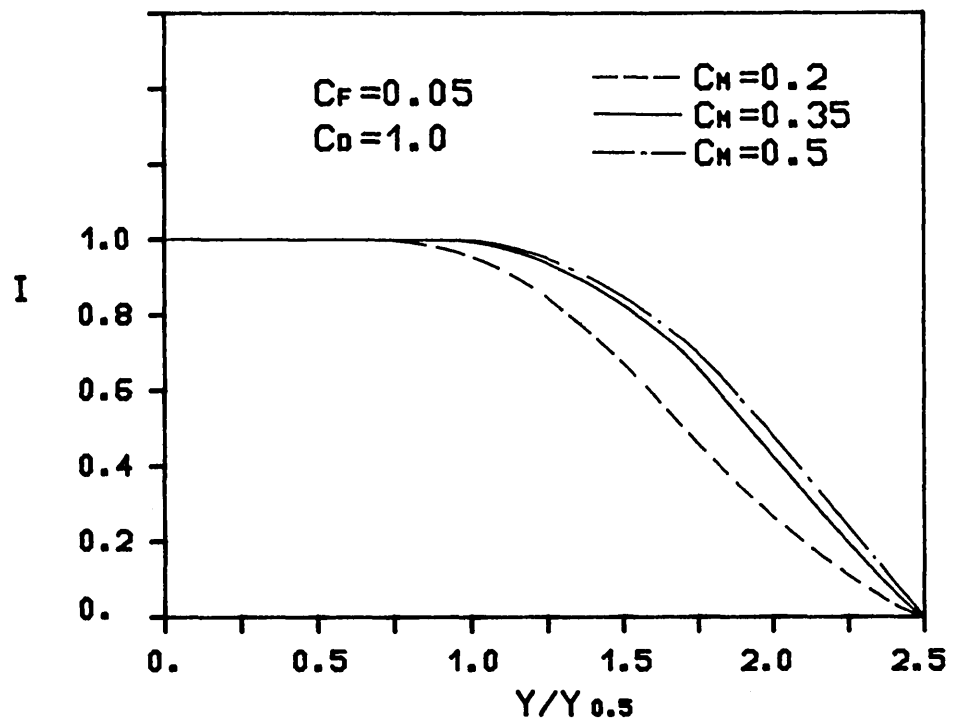


FIG. 4.5-3 EFFECT OF C_M ON THE PREDICTED SIMILARITY-PROFILE OF THE INTERMITTENCY FACTOR

Similarly, Figs. 4.5-4, 4.5-5, and 4.5-6 show the effects of C_f while Figs. 4.5-7, 4.5-8, and 4.5-9 show those of C_d .

4.6 Discussion

The simulation of an air-into-air steady jet has been described in this chapter. The predicted mean velocity along the centre-line (Fig.4.4-1), the streamwise velocity profiles of the turbulent, non-turbulent fluids, and the mean flow (Fig.4.4-2), and the profile of the intermittency factor (Fig.4.4-3) in the self-similar region show satisfactory agreements with the experimental data. The rate of spread (Table 4.4-1) is underpredicted by about 6%. Nevertheless, overall agreement is satisfactory. It confirms the suitability of the mathematical model described in Chapter 2 for calculating the air-into-air jet under steady-state conditions. These computational results will be compared with those of the transient jet simulated in the following chapter.

Spalding (1979) used the constant-effective-viscosity assumption with its empirical constant in a single-fluid mathematical model to predict successfully the mean flow of the air-into-air steady jet. In this chapter, the same constant-effective-viscosity assumption with the two-fluid assumptions was successfully used for the same purpose. With the two-fluid assumptions, however, the model can, in

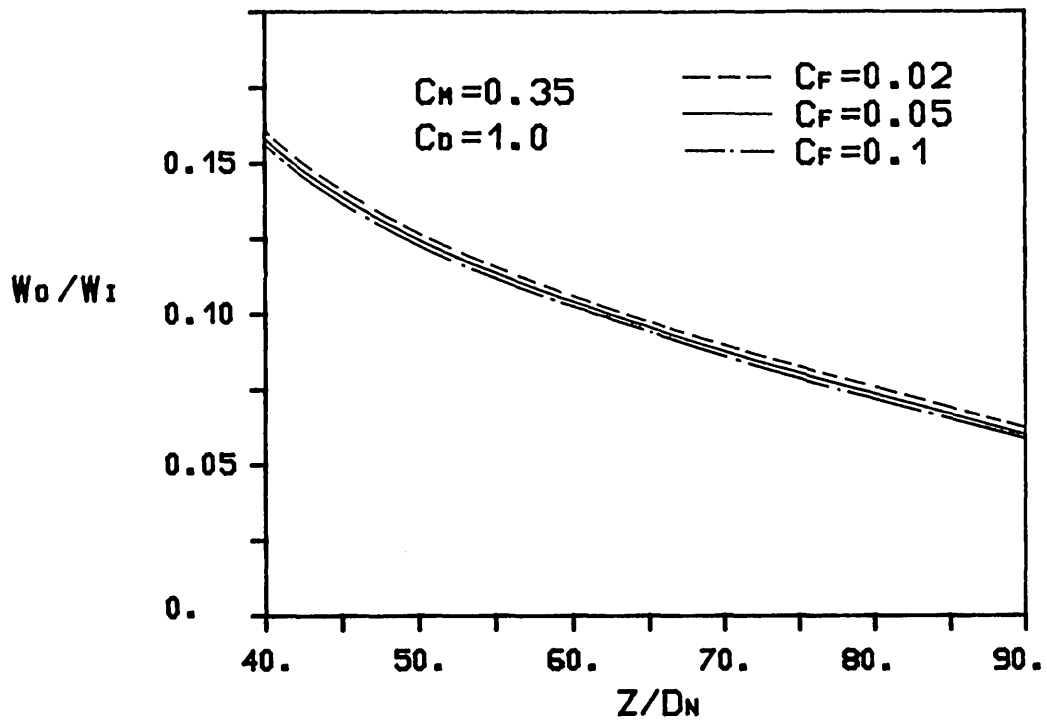


FIG. 4.5-4 EFFECT OF C_f ON THE MEAN VELOCITY ALONG THE CENTRE-LINE

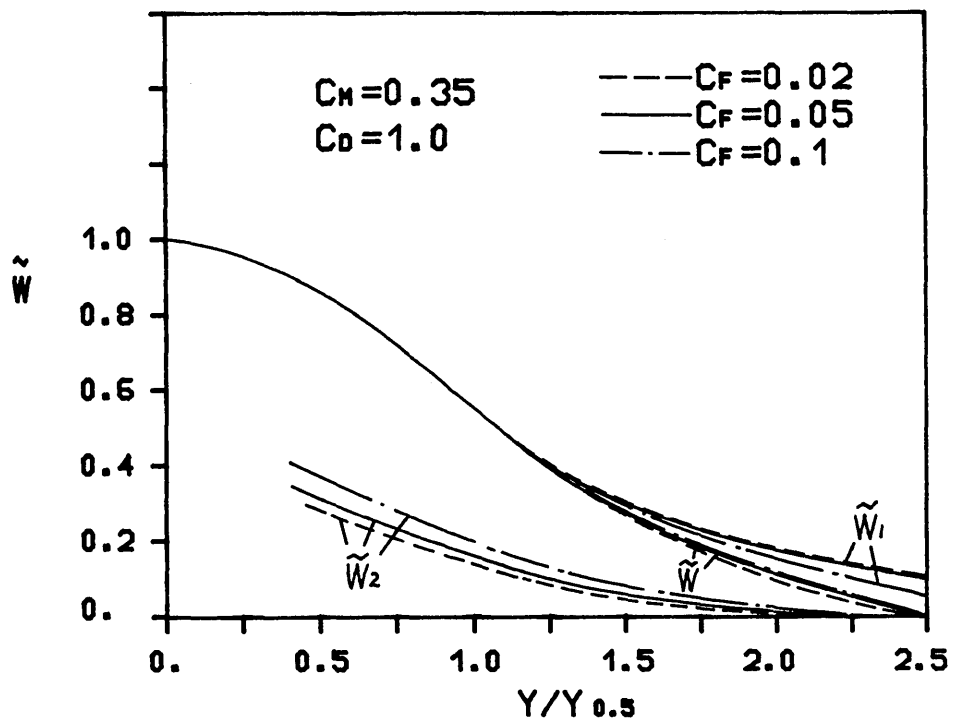


FIG. 4.5-5 EFFECT OF C_f ON THE PREDICTED SIMILARITY-PROFILES OF THE STREAMWISE VELOCITIES

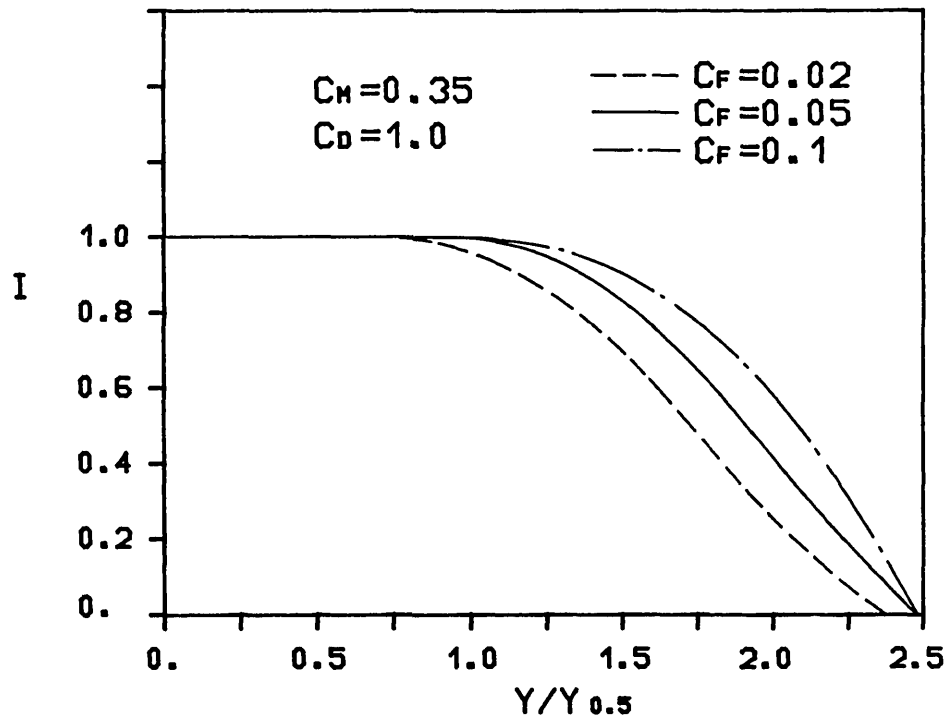


FIG. 4.5-6 EFFECT OF C_F ON THE PREDICTED SIMILARITY-PROFILE OF THE INTERMITTENCY FACTOR

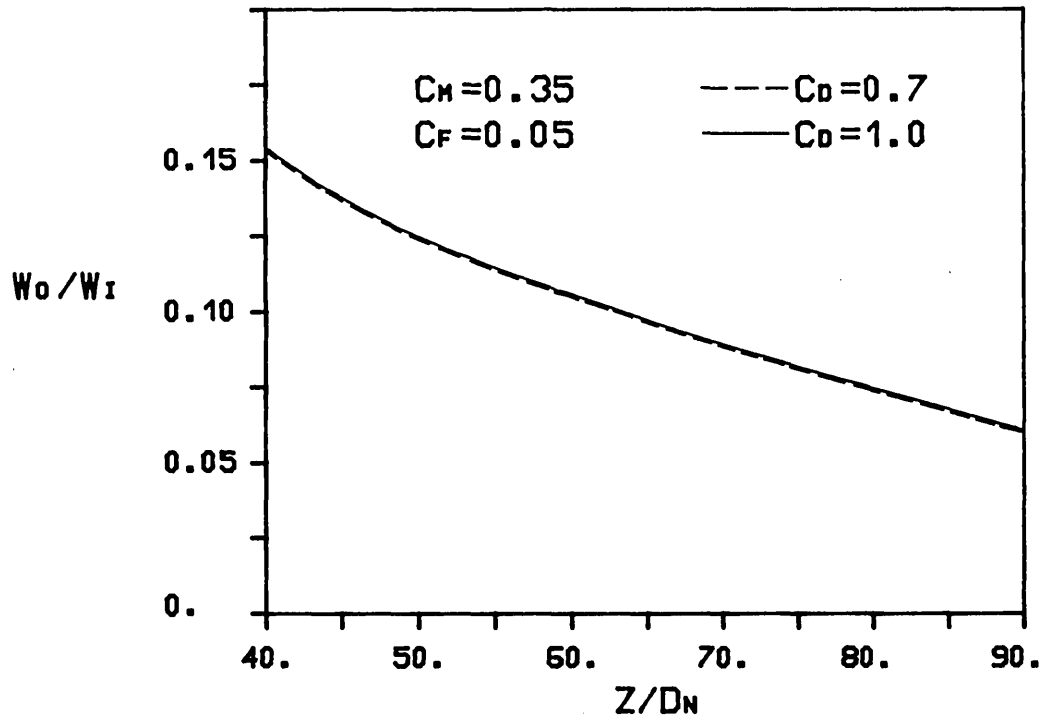


FIG. 4.5-7 EFFECT OF C_D ON THE MEAN VELOCITY ALONG THE CENTRE-LINE

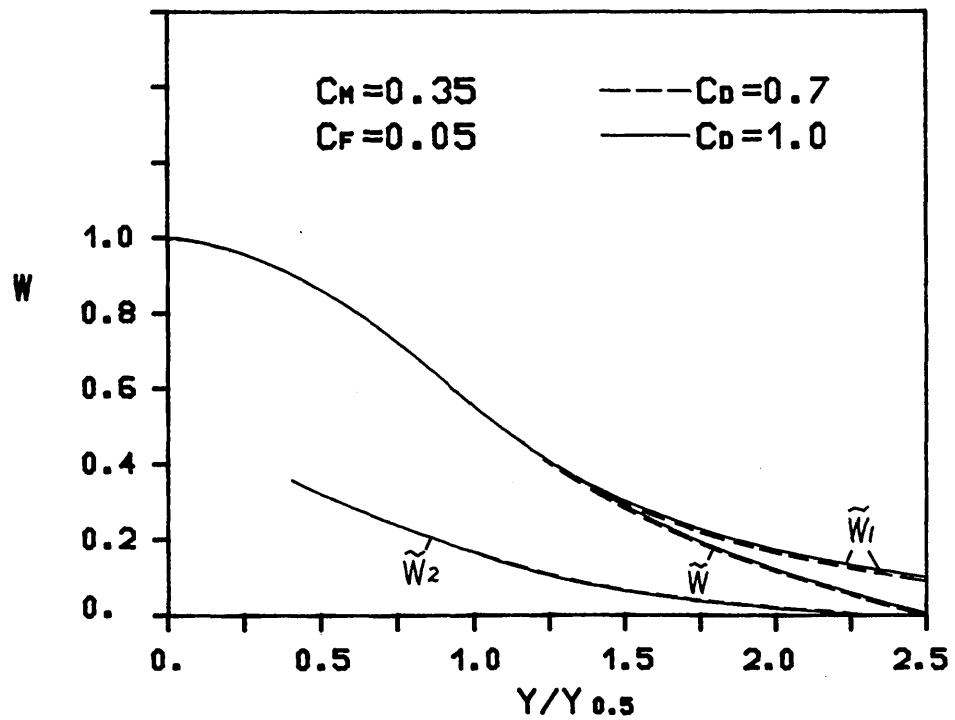


FIG. 4.5-8 EFFECT OF C_D ON THE PREDICTED SIMILARITY-PROFILES OF THE STREAMWISE VELOCITIES

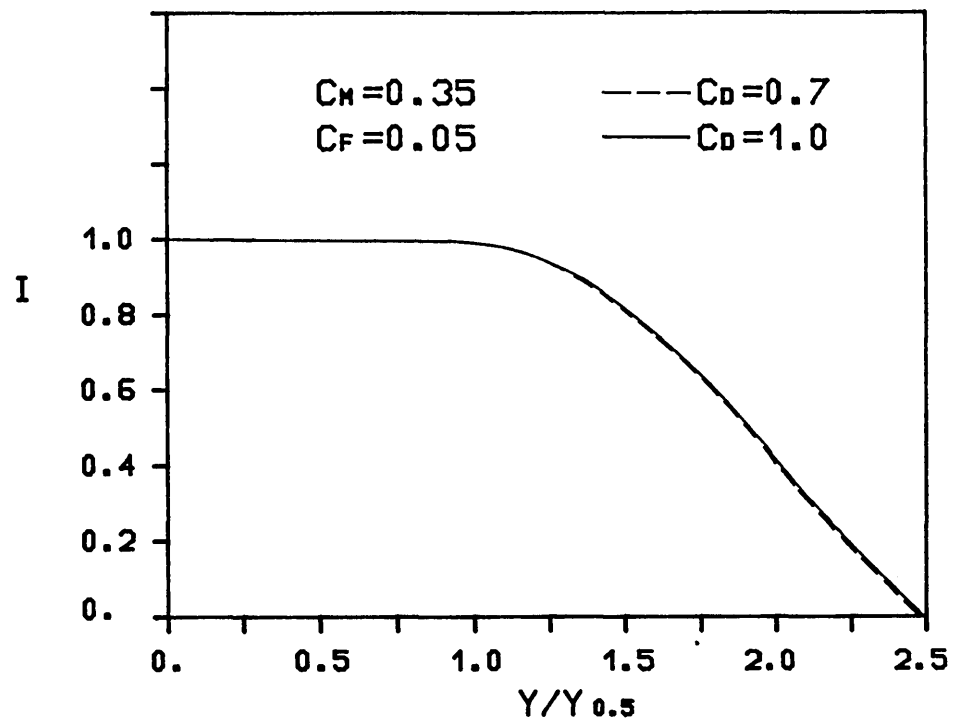


FIG. 4.5-9 EFFECT OF C_D ON THE PREDICTED SIMILARITY-PROFILE OF THE INTERMITTENCY FACTOR

addition, predict the streamwise velocity profiles of the turbulent and non-turbulent fluids and the profile of the intermittency factor. These intermittency phenomena of turbulent flows were observed in several experiments; but could not be predicted using conventional single-fluid turbulence model.

When the constant-effective-viscosity assumption is made, it has been found that the constants of the two-fluid assumptions have very little effect on the quantities of the mean flow such as the rate of spread (Table 4.5-1), the mean velocity along the centre-line (Figs.4.5-1, 4, 7), and the streamwise velocity profiles of the mean flow (Figs.4.5-2, 5, 8); but these constants have some small effect on the profiles of the streamwise velocity of the two fluids and the intermittency factor (Figs. 4.5-2, 3, 5, 6). All these effects were expected. Since the intermittency factor in most part of the flow is equal to 1 or close to 1 (Fig.4.4-3), the mean flow is dominated, to a great extent, by the turbulent fluid. The constant-effective-viscosity assumption gives the effective viscosity of the turbulent fluid. This assumption would, therefore, dominate the prediction of the mean flow, while the two-fluid assumptions can only be expected to have little effect on the mean flow, and have some effect on the profiles of the two fluids and intermittency factor.

In the present study, only some assumptions of the two-fluid model have been incorporated into the mathematical model. It should be noted that the two-fluid model itself is capable of predicting the jet flows without using the effective viscosity assumption. This can be done using the expression for the exchange coefficient Γ_ϕ , which has been proposed by Spalding (1985a) and Ilegbusy (1985). This particular possibility has not been fully tested by the present author; but could be a subject for future work.

4.7 Conclusion

The air-into-air steady jet has been simulated by using the mathematical model in which the constant-effective-viscosity model and some assumptions of the two-fluid model have been incorporated.

Comparisons between the computational and experimental results of the mean velocity along the centre-line, the streamwise velocity profiles of the two fluids and the mean flow, the profile of the intermittency factor, and the rate of spread in the self-similar region show satisfactory agreements.

The model and the assumptions employed are suitable for calculating the flow studied in this chapter. The recommended empirical constants are: $C_\mu=0.0142$, $C_m=0.35$,

$C_f=0.05$, and $C_d=1.0$.

The computed results will be compared with those of a transient jet.

As expected, the prediction of the mean flow has been dominated by the constant-effective-viscosity assumption; the two-fluid assumptions have demonstrated the intermittency phenomenon, but have very little effect on the mean flow.

CHAPTER 5 - COMPUTATION OF AN AIR-INTO-AIR TRANSIENT JET

5.1 Introduction

Having dealt with steady jets, attention will now be turned to the main subject of this thesis, viz. the investigation of transient jets. The first of them to be considered is an air-into-air transient jet. The behaviour of this transient jet is simulated in this chapter.

For the steady jet considered in the previous chapter, a steady-state condition was taken. For a transient jet, however, the flow is developing with time. It penetrates in the direction downstream of the nozzle.

The mathematical model for both the steady and transient jets under consideration was provided in Chapter 2. It is worth noting that the governing equations for a transient jet and those for a steady one are slightly different. The former differ in form from the latter in the possession of terms which express the time-dependent behaviour of the transient flow. By specifying appropriate boundary and initial conditions, the numerical solution procedure with the modelling assumptions embodied in the PHOENICS-84 computer code can be adapted to solving the governing equations of the transient jet considered in this chapter.

The process of the transient jet is of course different to that of the steady jet. In this study, however, the same modelling assumptions are applied to both steady and transient jets. This will be discussed later.

In the flow considered in this chapter, air is injected into air through a vertically-directed nozzle. The transient jet formed is a single-phase flow without buoyancy effects.

As reviewed in Chapter 1, this type of flow has not been much investigated in either experimental or theoretical ways. In this chapter, the constant-effective-viscosity model with some assumptions of the two-fluid model is again applied to this transient jet. Predictions of the mean velocity distribution along the centre-line varying with time, mean velocity developments and profiles of the streamwise velocities and intermittency at several given axial stations are all given. These computational results are compared with those of steady jet obtained in Chapter 4 and Ong's experimental data (1970).

In the following sections, Section 5.2 gives the flow conditions, Section 5.3 describes all the computational details, Section 5.4 presents the main results of the computations and comparisons, Section 5.5 demonstrates the

effect of the empirical constants of the two-fluid assumptions on the predictions, Section 5.6 is devoted to the discussion, followed by the conclusion at the end of this chapter.

5.2 The flow considered

A turbulent, round, air-into-air, transient jet is considered in this chapter.

Ong (1970) carried out experiments for this flow, and provided detailed experimental conditions and measurements. Comparisons of the computations are made with Ong's data.

The flow situation can be described by the following:-

- air is injected from the start;
- the surrounding air is initially at rest;
- the flow is transient;
- buoyancy is absent;
- axial symmetry prevails; and
- the flow is turbulent.

In the computations, the flow is presumed to be impulsively started, i.e. the nozzle velocity is reached instantaneously. In Ong's experiments, however, it took

from 2 ms. to 4 ms. for the different cases considered for the nozzle velocity to reach its steady-state value. Because this time is short in comparison with the duration of the whole process (60 ms. to 200 ms.), the "impulsive-starting condition" is assumed, in order to simplify the computation.

The conditions of the flows which are to be predicted are summarized in Table 5.2-1 below.

No. of case	Dia. of nozzle (Dn: mm)	Nozzle exit velocity (Wi: m/sec.)	Laminar Reynolds number (Re)
5-1	25.4	121.92	206,450
5-2	25.4	60.96	103,230
5-3	12.7	121.92	103,230

Table 5.2-1 Flow conditions considered

Equation (4.2-1) gives the definition of the Reynolds number.

5.3 Computational details

5.31 Solution procedure and calculation domain

The governing equations with the modelling assumptions for the transient flows were presented in Chapter 2. These equations are now numerically solved using the PHOENICS-84 computer code.

Different from the steady jet predicted in Chapter 4, the transient jet considered in this chapter is an elliptic and transient flow. Therefore, as the integration marches forward in time, additional storage is required for "old" as well as "new" variables. The computer storage and time for the present case is greatly increased.

As for the steady jet, fluid-1 refers to the turbulent (rotational) fluid and fluid-2 refers to the non-turbulent (irrotational) one.

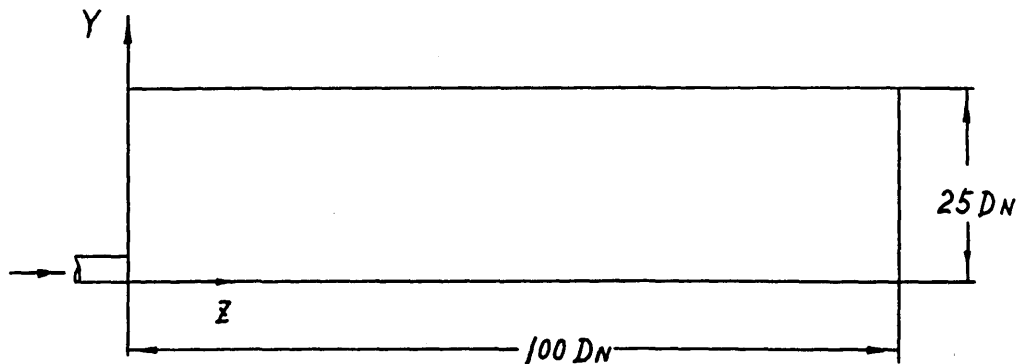


FIG. 5.31-1 THE CALCULATION DOMAIN FOR THE
AIR-INTO-AIR TRANSIENT JET

Fig. 5.31-1 sketches the integration domain for the calculation. The flow is symmetrical about the flow axis so that the calculation needs only to be carried out over one-half of the flow. The width of the domain is 25 nozzle diameters while the length is 100 nozzle diameters.

These distances along the two directions of the domain were found by Ong (1970) to be sufficiently remote from the transient jet flow field for the prescribed boundary conditions to be valid.

5.32 The grid and time-step

The algebraic finite-domain equations presented in Chapter 3 were derived from the governing partial-differential equations.

In order that the solution of the finite-domain equations become close approximation, the number of grid-nodes and time-steps should be large enough for any further increase in their number to effect only insignificant changes in the values of the important predicted quantities. Increasing the number of grid-nodes and time-steps also increases the computational expense. An investigation, therefore, has been carried out to determine the optimum number of grid-nodes and time-steps, giving a sufficiently accurate solution which is also economic.

On the other hand, numerical accuracy depends not only upon how many grid-nodes and time-steps are employed, but also upon how they are distributed. Non-uniformly distributed grid-nodes and time-steps were used in the present computations. Smaller intervals were set for the radial nodes near the axis, and for the axial nodes near the jet inlet while smaller time-steps were set during the initial period of the computations.

Case 5-1 was used as a test case. The results from a gradual increase in the number of grid-nodes and time-steps are now presented.

Grid

400, 625, 700, 800, and 875 grid-nodes, i.e. 20 x 20, 25 x 25, 25 x 28, 25 x 32, and 25 x 35 nodes in the radial and axial directions, have been tested.

The nodes were non-uniformly distributed. In the radial direction and from the axis outwards, the first 20% of the node occupied 2% of the domain, the next 40% of the node occupied 28% of the domain, and the remainder 40% of the node occupied the rest of the domain.

In the axial direction, the nodes were distributed according to the following power-law:

$$\frac{Z}{ZWLAST} = \left(\frac{IZ}{NZ} \right)^a \quad (5.32-1)$$

where Z is the axial distance of a node;
ZWLAST is the axial length of the domain;
IZ is the index of the node;
NZ is the total number of the axial nodes; and
a is an exponent.

a = 1.5 was adopted in the present computation.

With an increasing grid-node number and a constant time-step of 0.001 sec., Figs. 5.32-1 and 5.32-2 show the mean velocity along the centre-line and the mean velocity developments at given axial stations. It can be found that the grid comprising 25 x 32 grid-nodes in the radial and axial directions is the optimum choice.

Time-step

After obtaining the best selection of the grid-node number, attention was turned to a selection of time-step. Using 25x32 nodes, the effect of the time-step on the predictions is now investigated.

Constant time-steps of 0.004, 0.003, and 0.002 sec. and

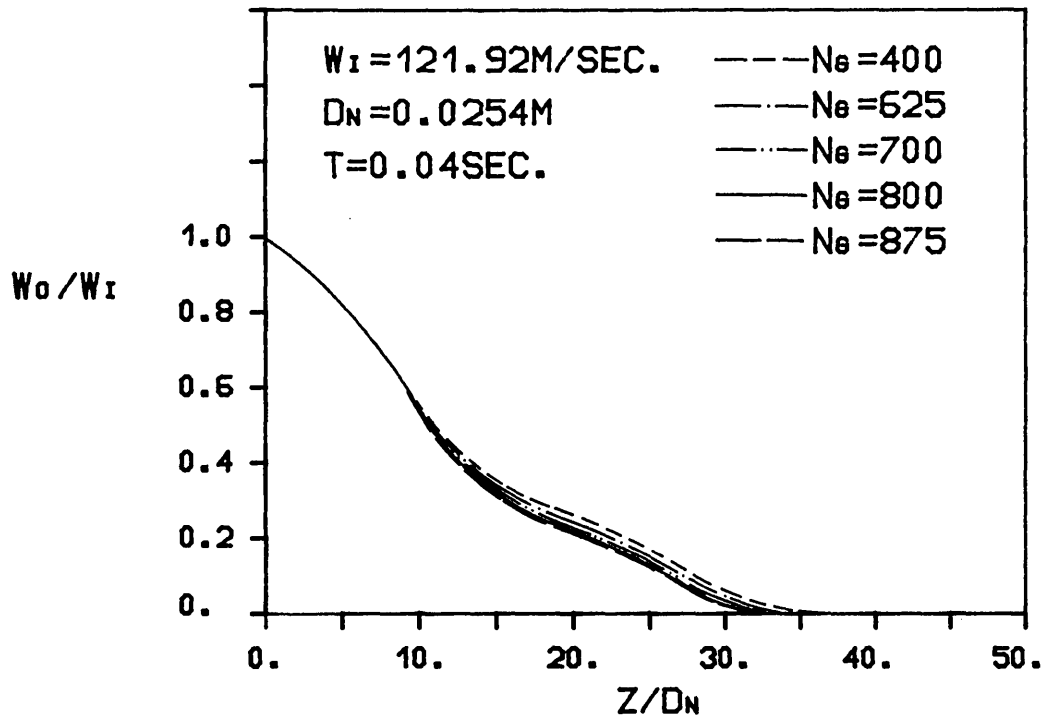


FIG. 5.32-1 EFFECT OF GRID-NODE NUMBER ON MEAN VELOCITY ALONG THE CENTRE-LINE

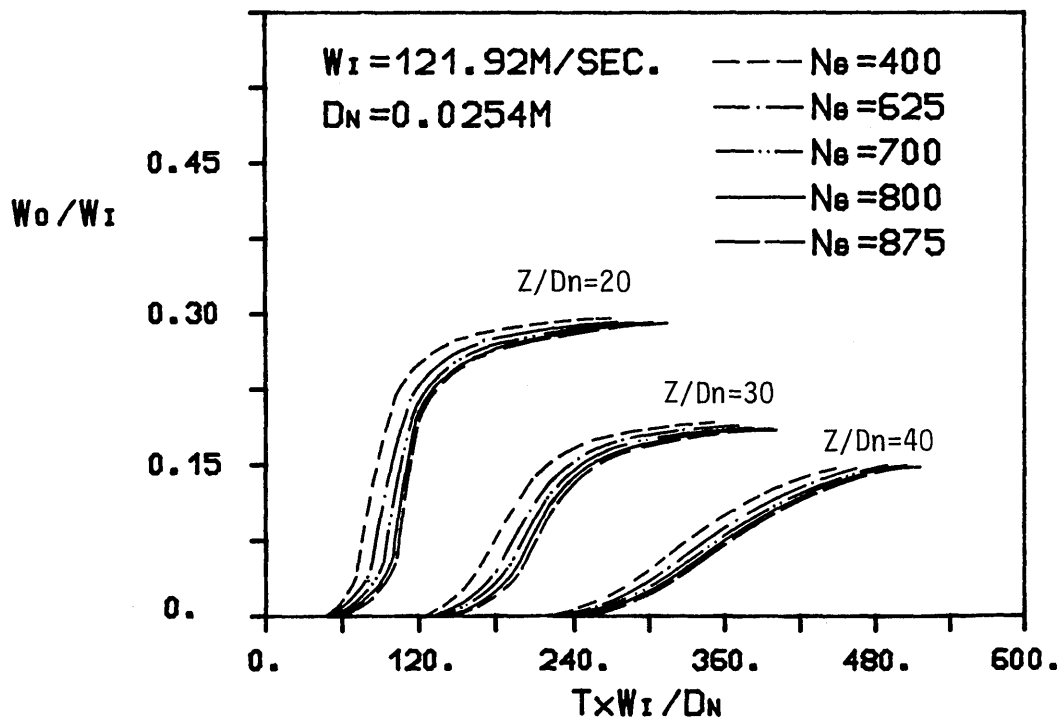


FIG. 5.32-2 EFFECT OF GRID-NODE NUMBER ON MEAN VELOCITY AT GIVEN AXIAL STATIONS

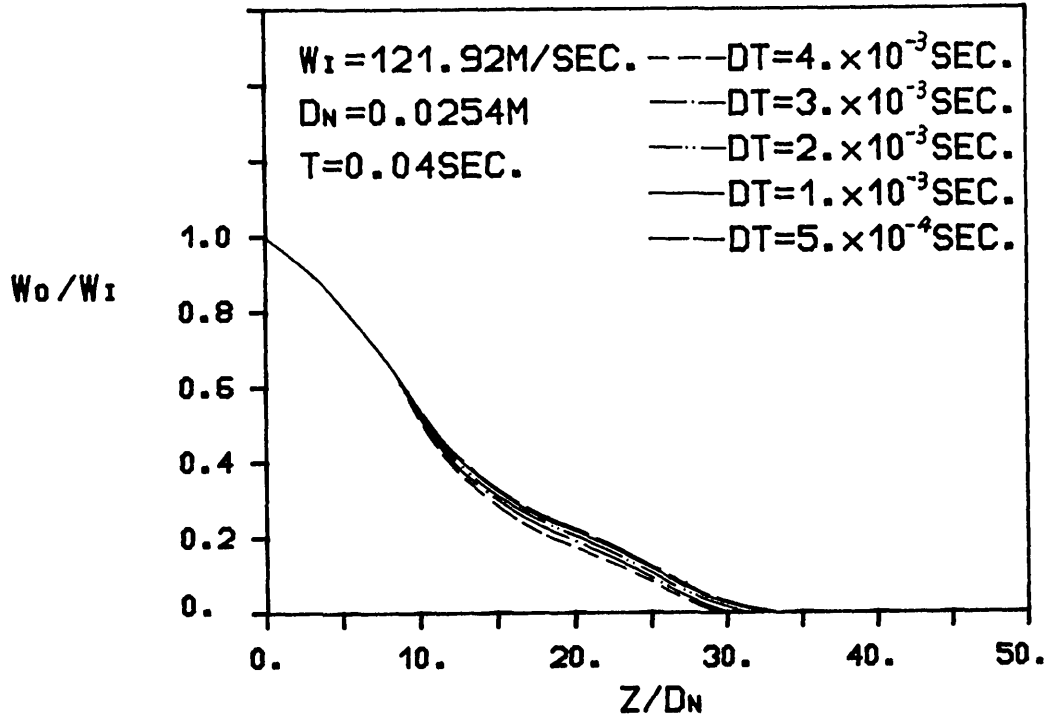


FIG. 5.32-3 EFFECT OF TIME-STEP ON MEAN VELOCITY ALONG THE CENTRE-LINE

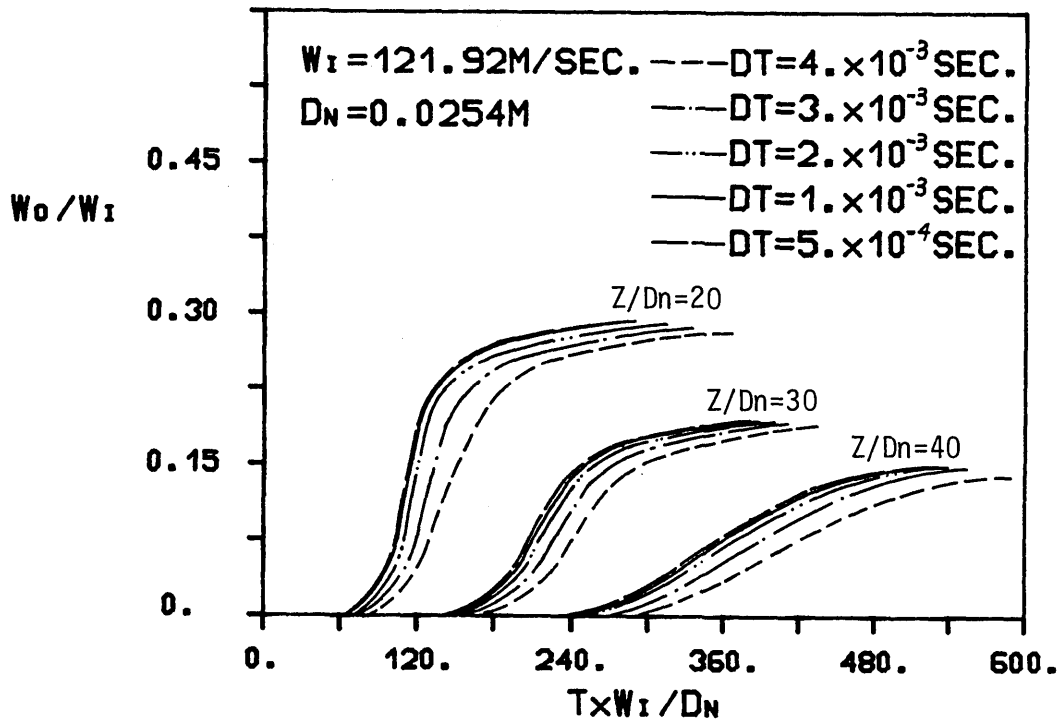


FIG. 5.32-4 EFFECT OF TIME-STEP ON MEAN VELOCITY AT GIVEN AXIAL STATIONS

expanding time-steps have been tested. For the latter, the time-step was initially given a small value of 1.0×10^{-5} sec., this was then increased by a factor 1.1 at every forward step until the assigned value of either 0.001 or 5.0×10^{-4} sec. was reached. The computed results of the mean velocity along the centre-line and the mean velocity developments at the given axial stations are shown in Figs. 5.32-3 and 5.32-4. In these figures, $DT=1 \times 10^{-3}$ sec. and $DT=5 \times 10^{-4}$ sec. refer to the expanding time-steps. It can be seen that the expanding time-steps from 1.0×10^{-5} sec. to the maximum of 0.001 sec. are considered to be the best choice.

The 25×32 nodes and expanding time-steps from 1.0×10^{-5} to the maximum of 0.001 sec. were used in producing the main results of this chapter.

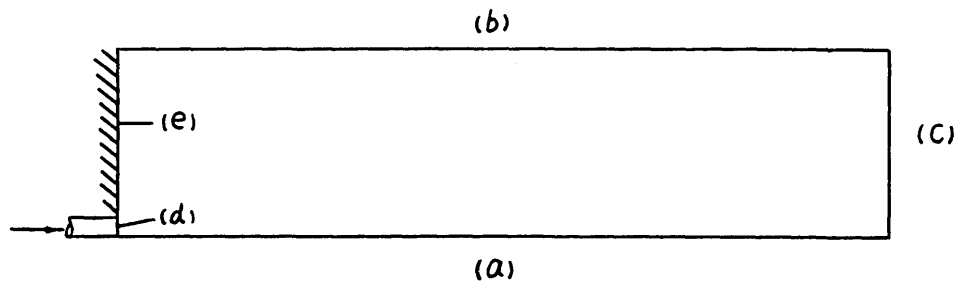


FIG. 5.33-1 THE BOUNDARIES OF THE CALCULATION DOMAIN FOR THE AIR-INTO-AIR TRANSIENT JET

5.33 Boundary conditions

The boundaries of the calculation domain are sketched in Fig. 5.33-1, and they are specified as follows:

- Boundary (a) is the inner boundary. This is a symmetrical axis, and consequently a zero-flux boundary condition is specified.

- Boundaries (b) and (c) are the outer and downstream boundaries of the domain respectively. Fixed-pressure boundary conditions are applied to these boundaries, i.e. the pressure as well as the velocity components are taken to be those of the surrounding air.

At boundary (d), i.e. the jet inlet,

- only the turbulent fluid is present, i.e. the intermittency factor is equal to 1;
- top-hat profiles are specified for the streamwise velocity of the turbulent fluid and the intermittency factor;
- no radial velocity component exists; and
- the streamwise velocity of the turbulent fluid is equal to that of the mean flow.

At boundary (e),

- a radial wall is present to simulate Ong's experimental condition. The wall function built into the code is directly applied. In this wall function, the Reynolds number is first evaluated, based on the resultant velocity, on the distance from the wall to the grid node, and on the density. If this is less than 132.25, a laminar wall function is used. If the Reynolds number turns out to be greater than 132.25, the presumed velocity variation is logarithmic, and the corresponding shear-stress coefficient is evaluated. This corresponds to the common-used "log-law" wall function. The detailed description of this function can be found in the reference of Rosten and Spalding (1984).

5.34 Initial conditions

Since the flow is transient, and it starts from rest, zero pressure, zero velocity components of the two fluids, $r_1 = 0$, and $r_2 = 1$ are specified over the whole domain.

5.35 Convergence

In order to obtain a converged solution, the relaxation facilities built in the PHOENICS-84 computer code, "FALSDT" and "LINRLX", were used. The typical value of the relaxation factor was 0.1.

With appropriate relaxation factors, 15 sweeps were required at each time step. At the end of each time step, the whole-field sum of the absolute value of the volume flow residuals was of the order of 10^{-4} .

Table 5.35-1 summarizes all the information about the convergence of the computations.

Detailed inputs into the PHOENICS-84 computer code can be found in the listings of the "Q1" and "GROUND" files in Appendix A.

Variable	Value
Grid	25 grid-nodes in Y-direction 32 grid-nodes in Z-direction
Type of grid spacing	non-uniform; finer near the axis and inlet
Time-step	expanding; from 1.0×10^{-5} to 1.0×10^{-3} by factor 1.1
Number of iterative sweeps at each time-step	15
Relaxation	FALSDT: 0.1 for V_1 , V_2 , W_1 , W_2
	LINRLX: 0.1 for r_1 , r_2
Whole-field sum of ABS(vol. flow residuals)	2.449×10^{-4}

Table 5.35-1 Computational details for the air-into-air transient jet

5.36 Computer storage and computer time

In this section, the computational requirements for the

present computations are presented.

Since the flow evolves parabolically in time, storage is required for both previous and current time steps. These computer storage and computer time required are functions of the number of grid-node and time-step used.

Table 5.36-1 displays the essential incore space and the computer time for the different grid-nodes used for whole domain.

Grid NYXNZ	Computer storage	Computer time per time-step (sec.)
20x20	8,845	84.5
25x25	13,070	131.97
25x28	14,378	147.88
25x32	16,097	156.0
25x32	17,405	171.29

Table 5.36-1 Computational requirement for the
air-into-air transient jet
(Computer used: Perkin Elmer
3220)

5.4 Main results

In this section, the results from the computations are presented. They are also compared with those for a steady jet and Ong's experimental data (1970).

Table 5.2-1 gives the flow parameters for three cases which have been computed. The results obtained are given bellow.

(1) Variations with time of the mean velocity along the centre-line

The variations with time of the mean velocity along the centre-line of case 5-1 are presented in Fig. 5.4-1. In this figure, the dimensionless variables W_o/W_i , Z/D_n , and TxW_i/D_n denote respectively the mean velocity along the centre-line, axial distance from the nozzle exit, and the elapsed time. Three curves in this figure represent the mean velocity distributions at $TxW_i/D_n = 200$, 300, and 400, corresponding to $t = 0.041$, 0.062,, and 0.083 sec. respectively.

The prediction of the mean velocity along the centre-line of the steady jet obtained in Chaper 4 is also shown in this figure.

In figure 5.4-1, it can be seen, that the transient jet

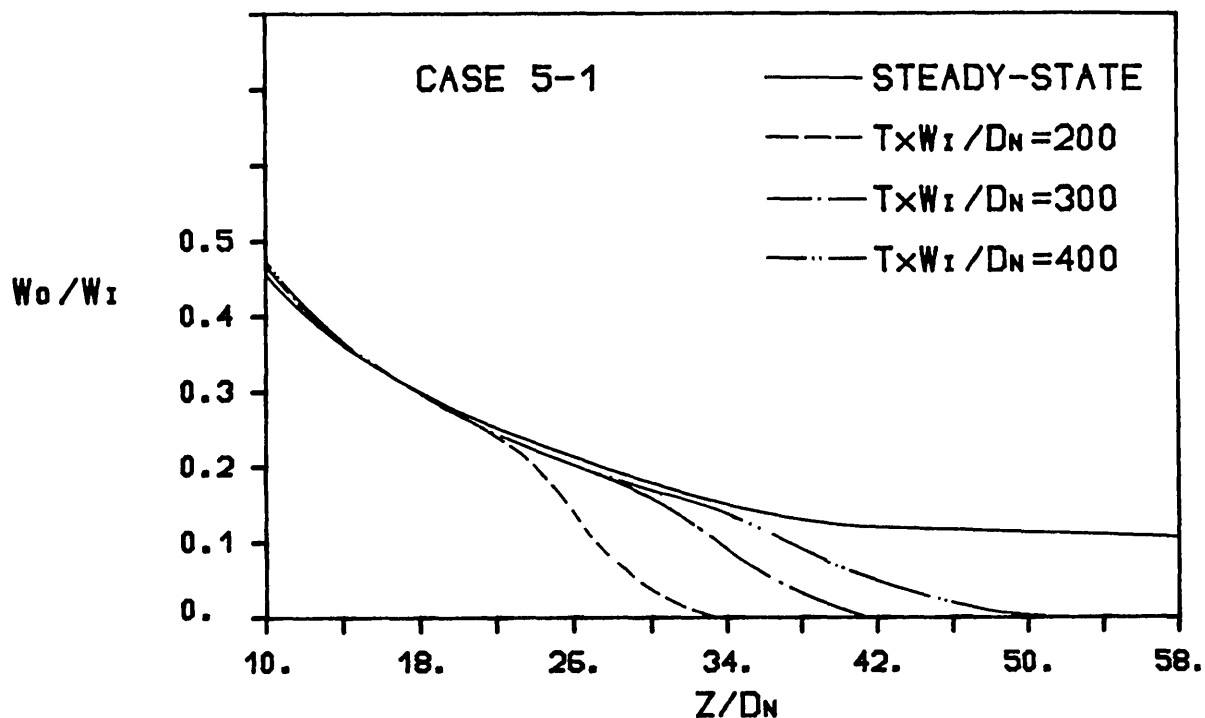


FIG. 5.4-1 VARIATION OF MEAN VELOCITY ALONG THE CENTRE-LINE WITH TIME

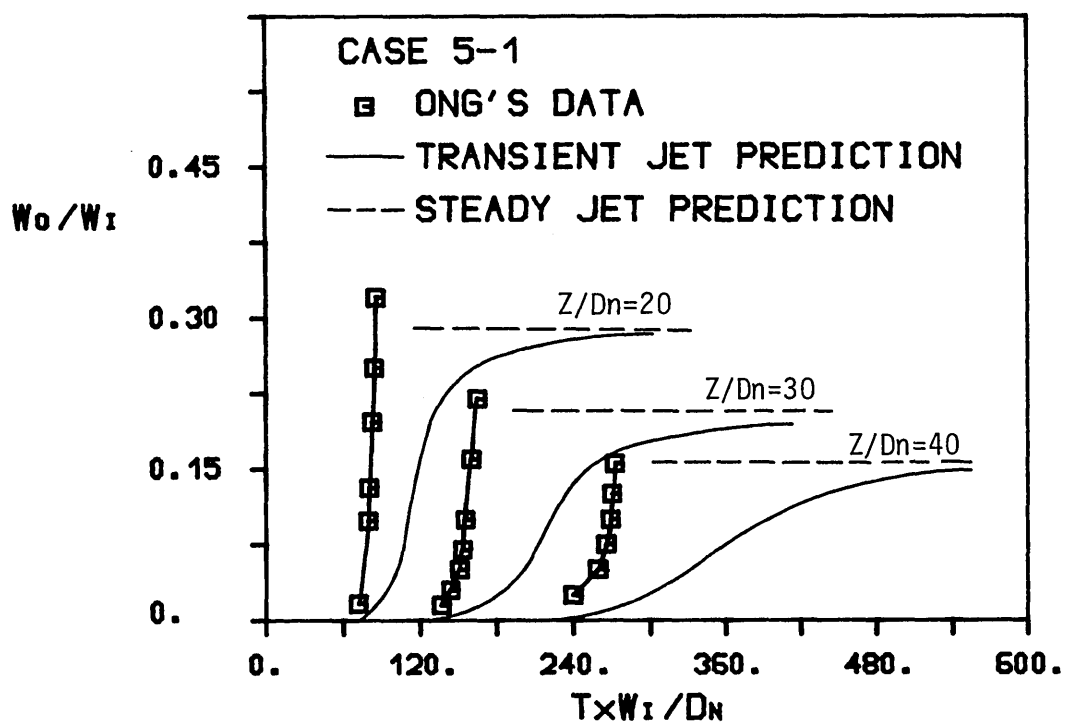


FIG. 5.4-2 DEVELOPMENT OF MEAN VELOCITY AT GIVEN AXIAL STATIONS: CASE 5-1

and further

penetrates further ^{and further} into the undisturbed region. Also the mean velocity at each axial station reaches a steady-state after certain time.

(2) Mean velocity developments at given axial stations

The predictions of the mean velocity developments with time at given axial stations are presented in Figs. 5.4-2, 5.4-3, and 5.4-4 for the three cases of Table 5.2-1. The axial stations for these computations are located at $z/D_n=20$, 30, and 40. Ong's experimental data are compared in these figures.

The predictions of the mean velocity at these stations for the steady jet are also presented in these figures.

It can be found from these figures that there is always a time-delay for the tip of a transient jet travelling from the nozzle exit to a certain axial station. As expected, the further the station is, the longer the time-delay is.

It can also be found that after a comparatively slow increase at early stage and a rapid increase afterwards, the mean velocity at each station settles gradually to a constant value. The closer the station is to the nozzle exit, the quicker it attains a constant velocity.

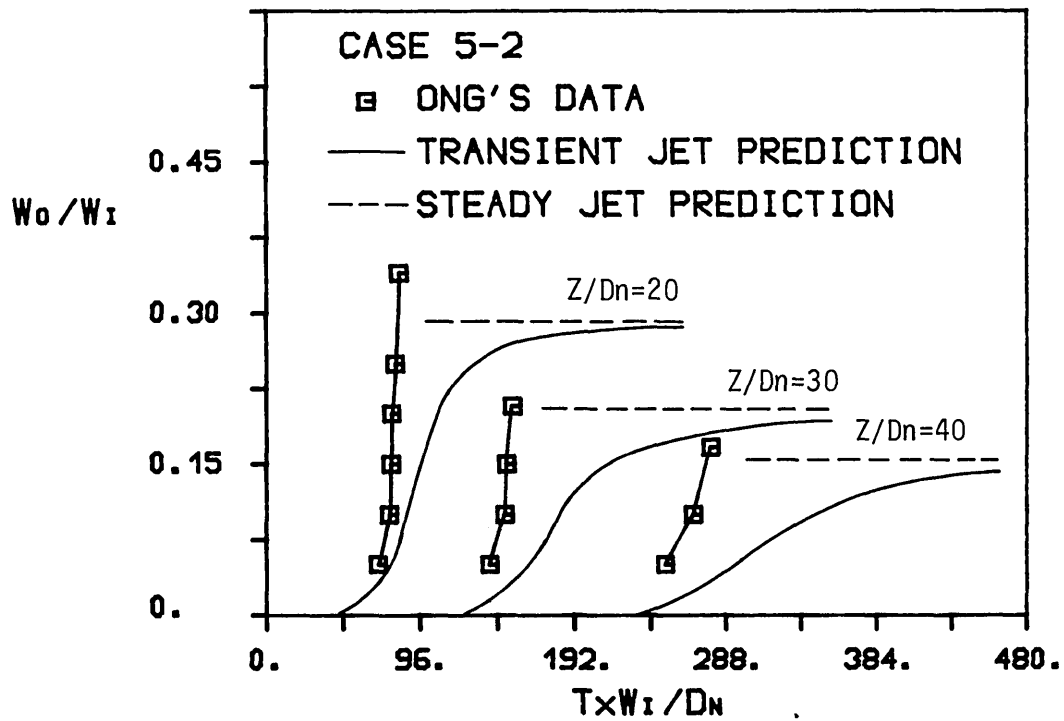


FIG. 5.4-3 DEVELOPMENT OF MEAN VELOCITY AT GIVEN AXIAL STATIONS: CASE 5-2

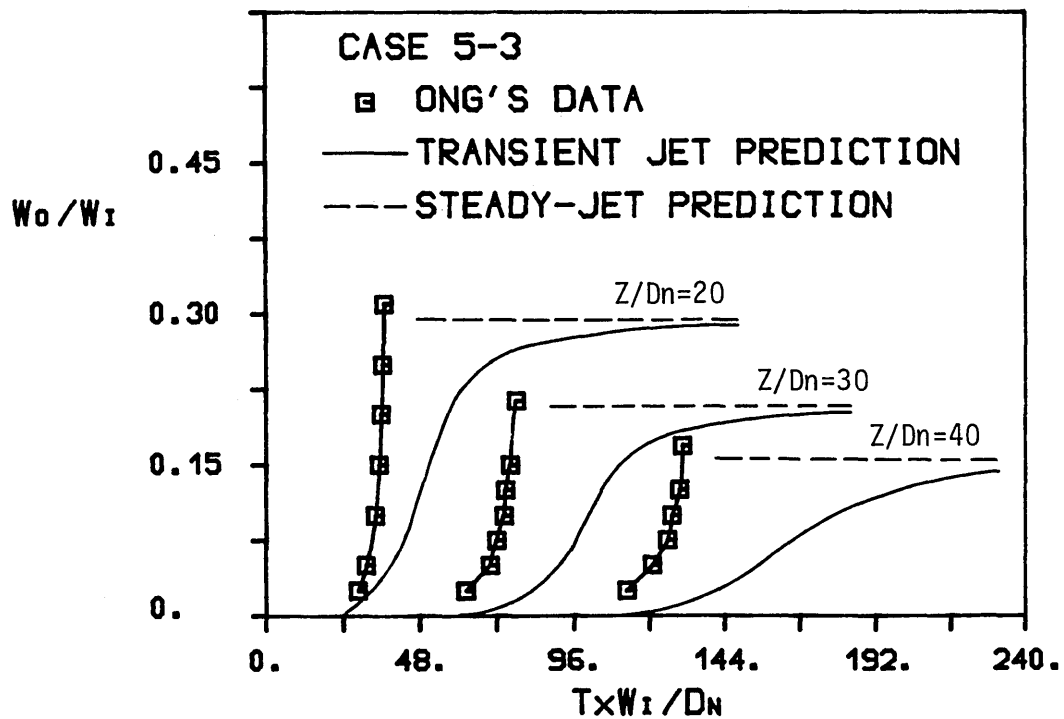


FIG. 5.4-4 DEVELOPMENT OF MEAN VELOCITY AT GIVEN AXIAL STATIONS: CASE 5-3

(3) Profiles of streamwise velocities and intermittency factor

Figs. 5.4-5 and 5.4-6 give, for case 5-1, the predicted profiles of the streamwise velocities of the two fluids, the mean flow, and the intermittency factor at $Z/D_n=20$, 30, and 40 and $t=0.11\text{sec.}$

Figs. 5.4-7 and 5.4-8 show, for case 5-1, the predicted profiles of the transient and steady jets at $Z/D_n=40$ and $t=0.11\text{sec.}$ The predictions for the steady jet were obtained in Chapter 4.

5.5 Effect of empirical constants of two-fluid assumptions

The calculations have been carried out in such a way, that the three constants of the two-fluid assumptions are varied individually.

The values of the constants which have been tested in the calculation are:

$$C_m = 0.2, \underline{0.35}, \text{ and } 0.5;$$

$$C_f = 0.01, \underline{0.05}, \text{ and } 0.1; \text{ and}$$

$$C_d = 0.7 \text{ and } \underline{1.0}.$$

where the underlined values are the standard ones.

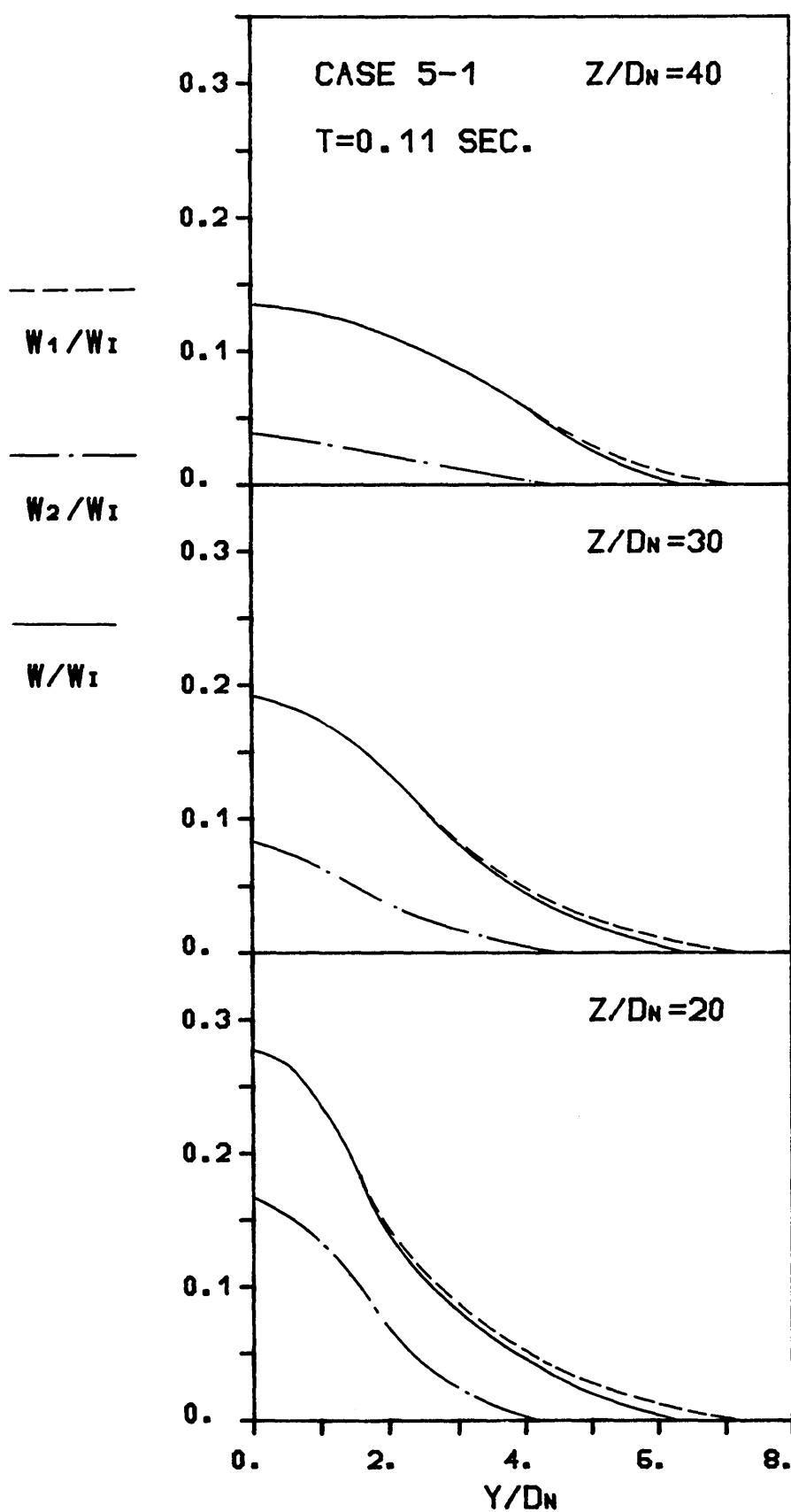


FIG. 5.4-5 PREDICTED PROFILES OF
 STREAMWISE VELOCITIES

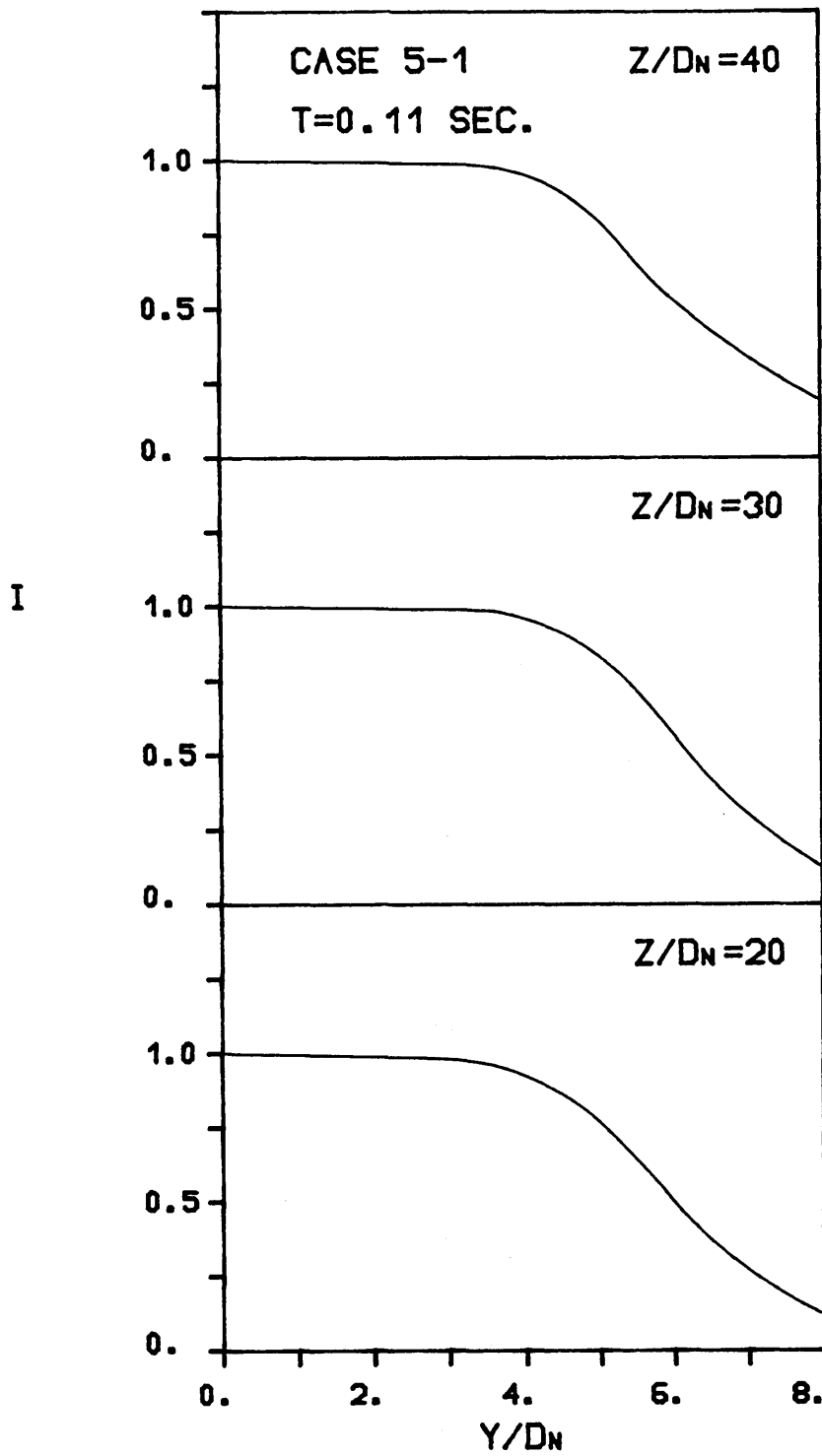


FIG. 5.4-6 PREDICTED PROFILES OF
INTERMITTENCY FACTOR

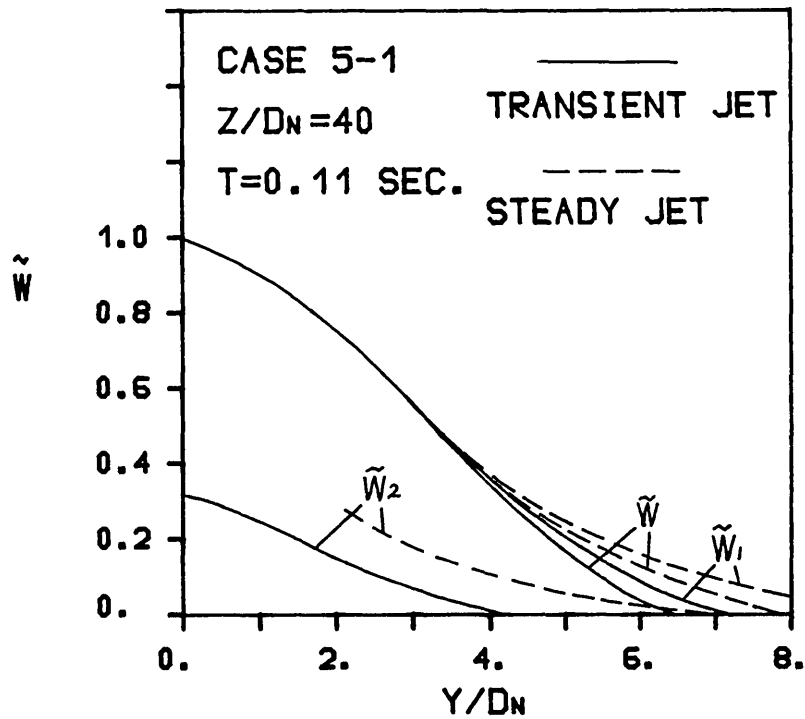


FIG. 5.4-7 PREDICTED PROFILES OF
 STREAMWISE VELOCITIES

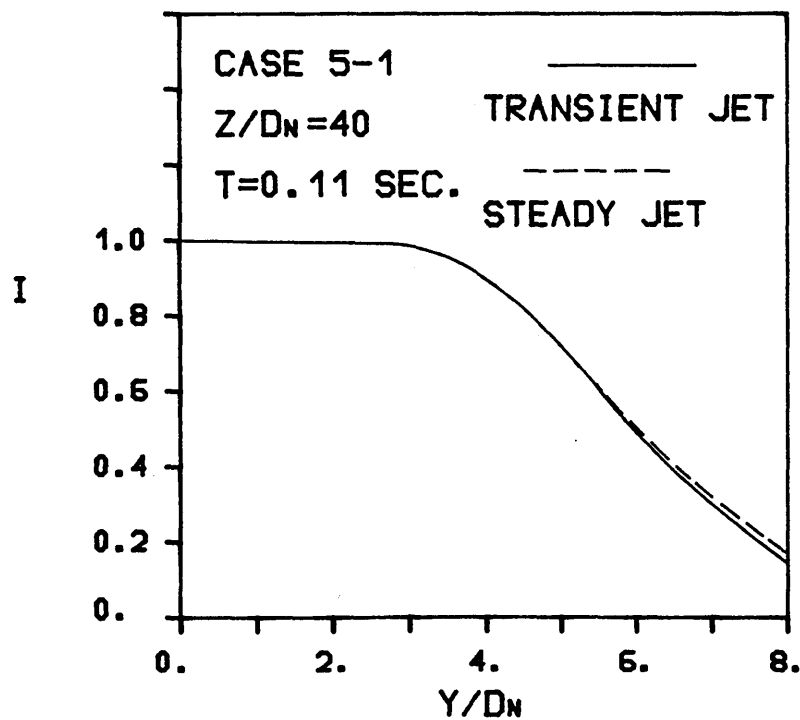


FIG. 5.4-8 PREDICTED PROFILES OF
 INTERMITTENCY FACTOR

The calculations were carried out for Case 5-1. The following results are adopted to show the effects of the empirical constants on: (a) the mean velocity along the centre-line at $t=0.04$ sec.; (b) the developments of mean velocity at $Z/D_n=20, 30,$ and 40 ; (c) the profiles of streamwise velocities at $Z/D_n=20$ and $t=0.04$ sec.; and (d) the profile of intermittency factor at $Z/D_n=20$ and $t=0.04$ sec..

Figs. 5.5-1 to 5.5-4 show the effects of C_m on the predictions; Figs. 5.5-5 to 5.5-8 show those of C_f , while Figs. 5.5-9 to 5.5-12 show those of C_d .

5.6 Discussion

In Fig. 5.4-1, it has been found that the predicted mean velocity along the centre-line of the transient jet attains that of the steady jet as expected. But slight differences still persist. For $Z/D_n < 18$, the mean velocity of the transient jet, when it reaches a steady state, is higher than that of the steady jet, while for $Z/D_n > 18$, it is lower. This discrepancy may be due to the difference between their integration domains. The domain for the steady jet is expanding in the radial direction while the width of the domain for the transient jet is a constant. Boundary conditions specified for the outer boundaries of these two domains are the same fixed-

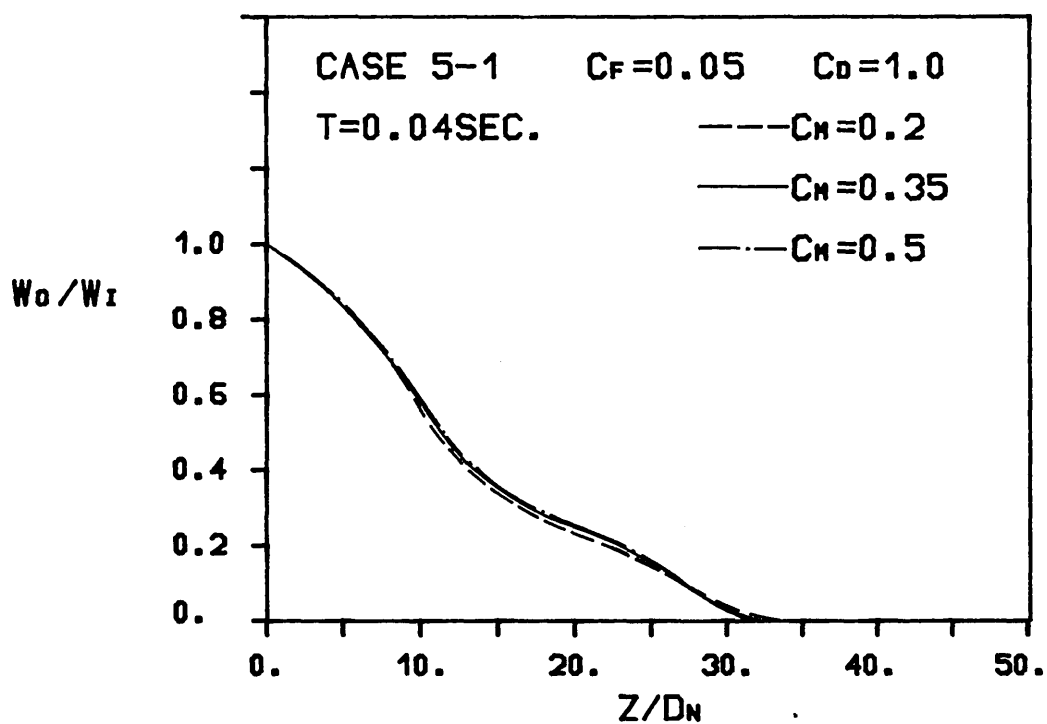


FIG. 5.5-1 EFFECT OF C_H ON MEAN VELOCITY ALONG THE CENTRE-LINE

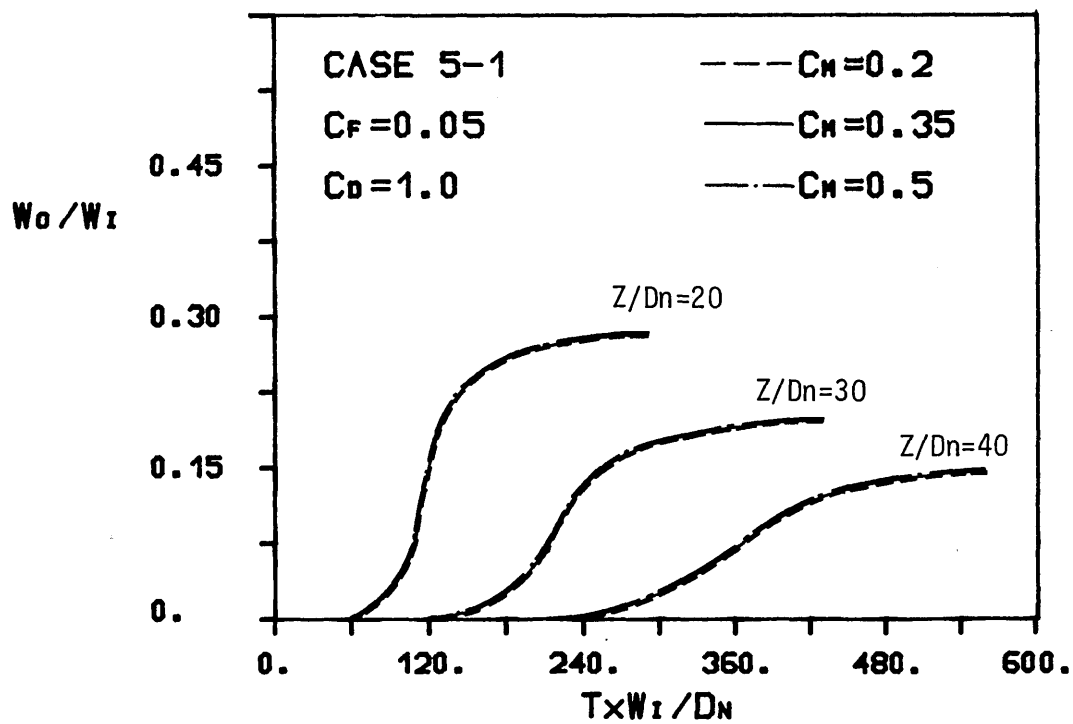


FIG. 5.5-2 EFFECT OF C_H ON MEAN VELOCITY AT GIVEN AXIAL STATIONS

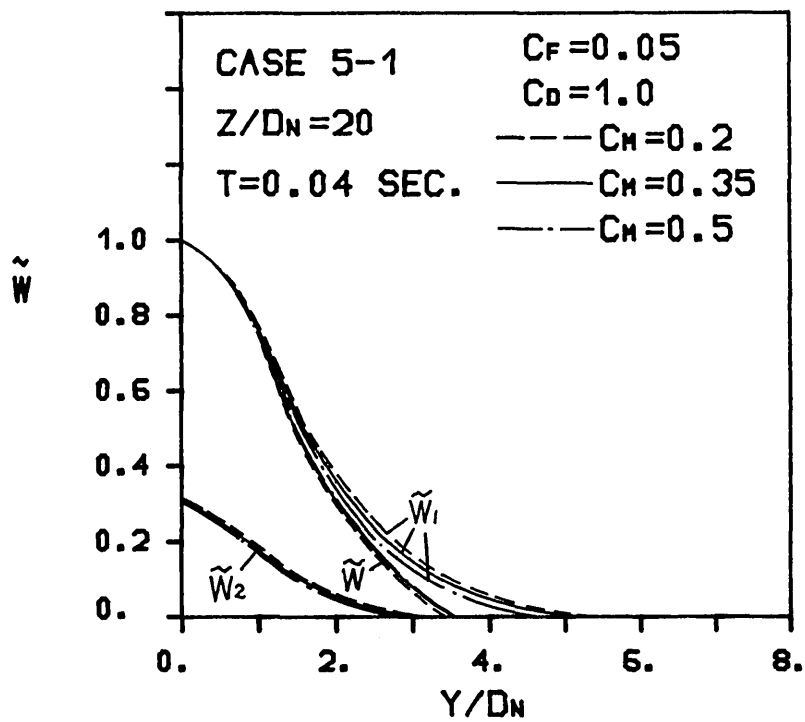


FIG. 5.5-3 EFFECT OF C_M ON THE PREDICTED PROFILES OF STREAMWISE VELOCITIES

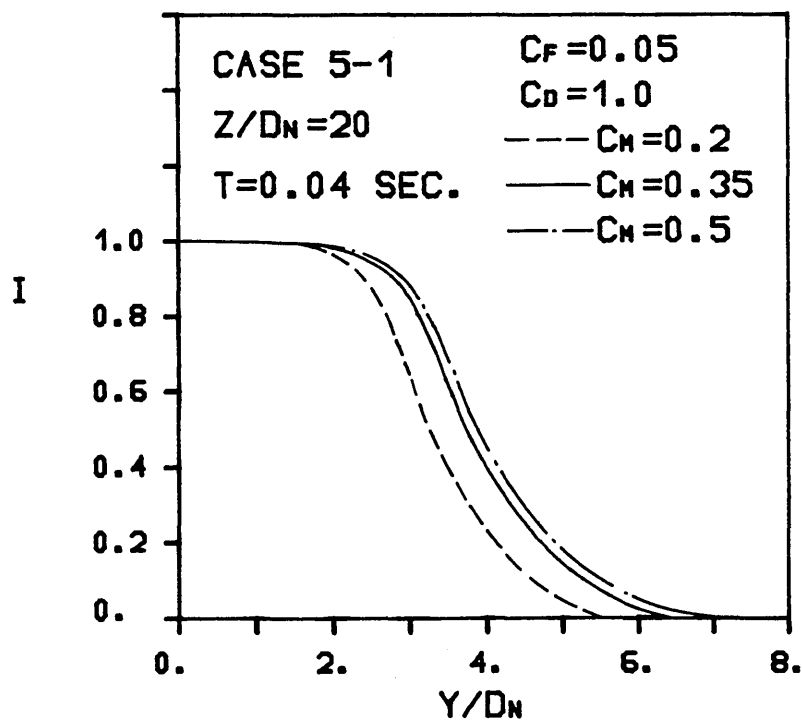


FIG. 5.5-4 EFFECT OF C_M ON THE PREDICTED PROFILES OF INTERMITTENCY FACTOR

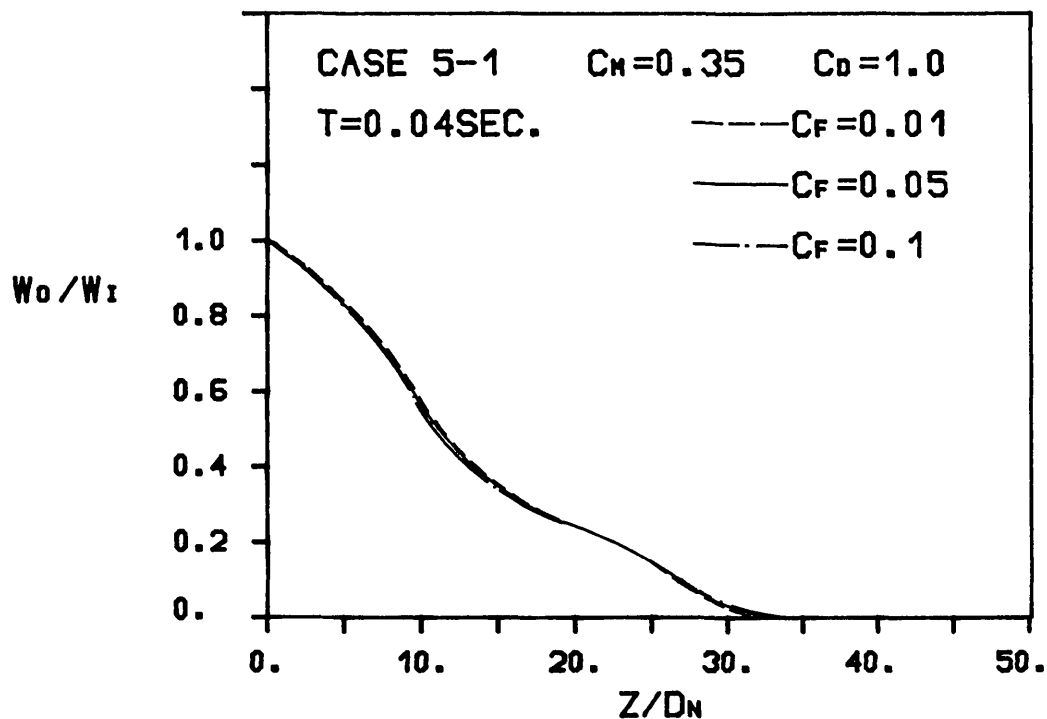


FIG. 5.5-5 EFFECT OF C_F ON MEAN VELOCITY ALONG THE CENTRE-LINE

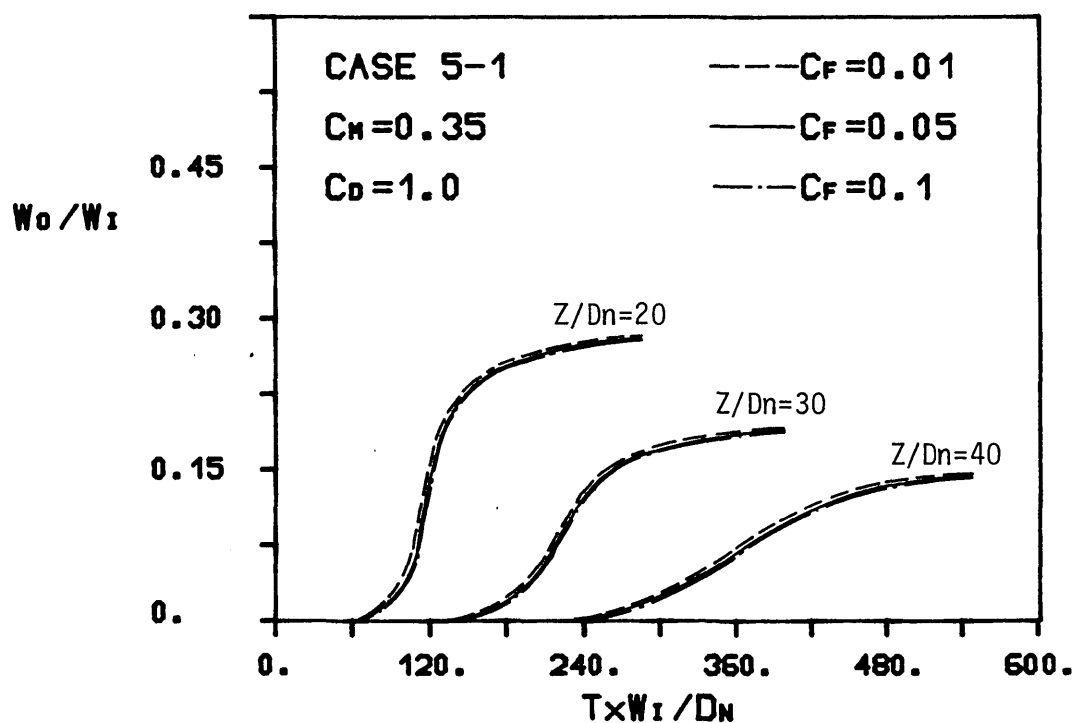


FIG. 5.5-6 EFFECT OF C_F ON MEAN VELOCITY AT GIVEN AXIAL STATIONS

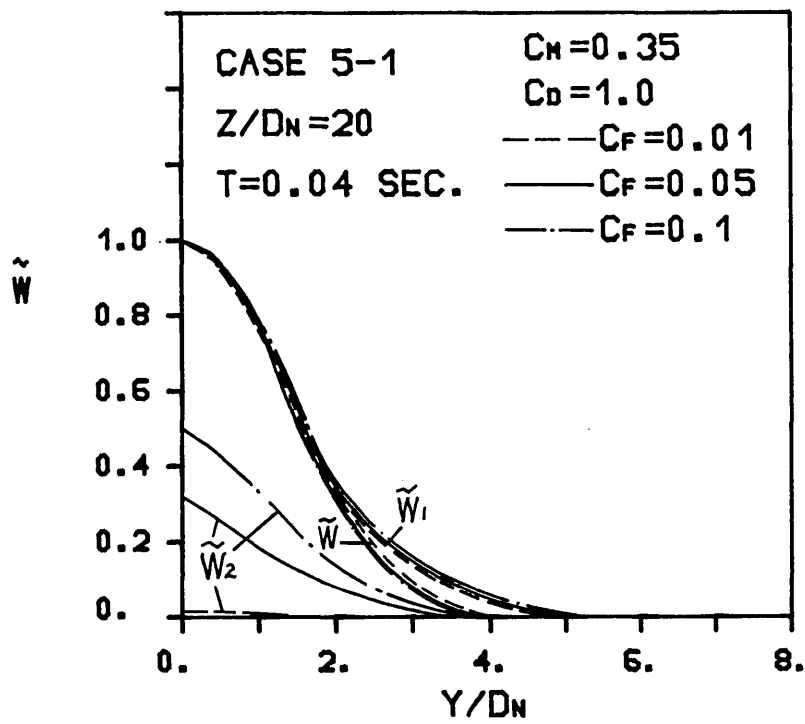


FIG. 5.5-7 EFFECT OF C_F ON THE PREDICTED PROFILES OF STREAMWISE VELOCITIES

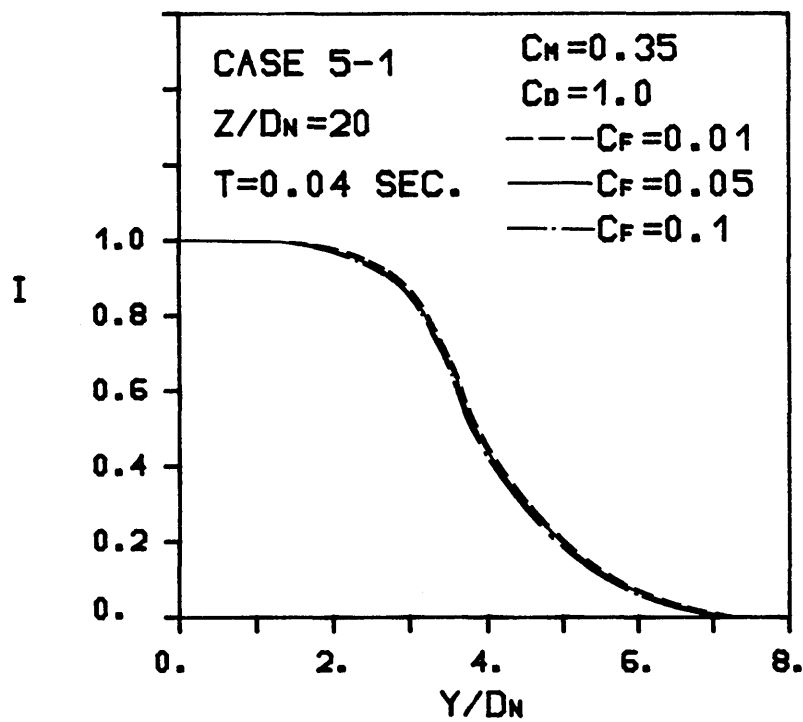


FIG. 5.5-8 EFFECT OF C_F ON THE PREDICTED PROFILES OF INTERMITTENCY FACTOR

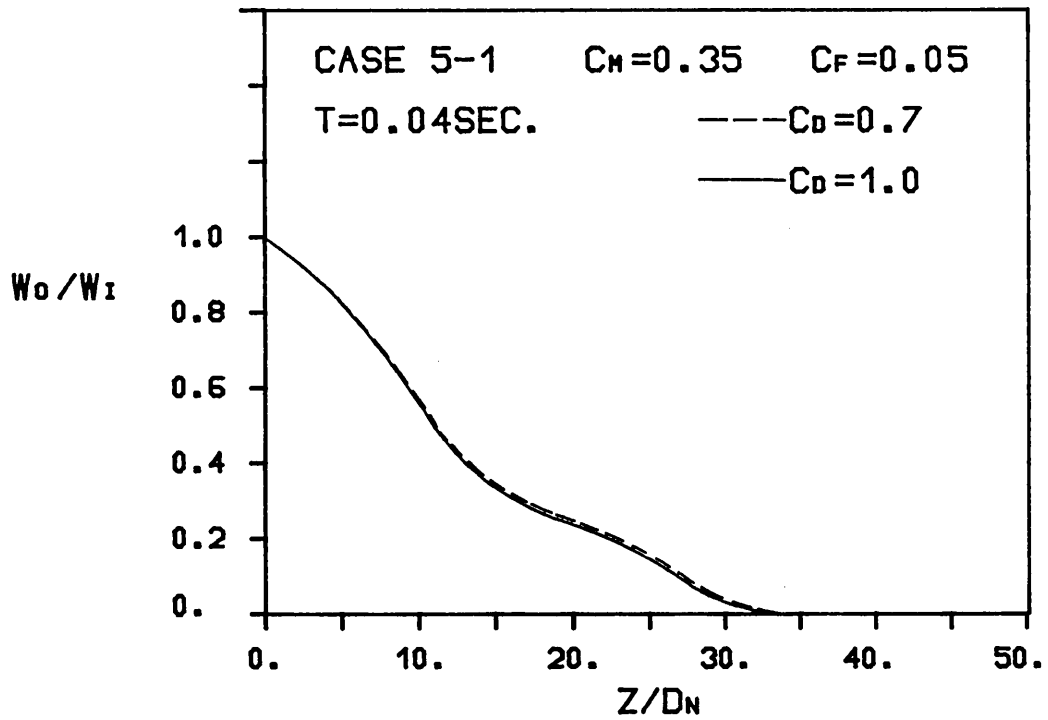


FIG. 5.5-9 EFFECT OF C_D ON MEAN VELOCITY ALONG THE CENTRE-LINE

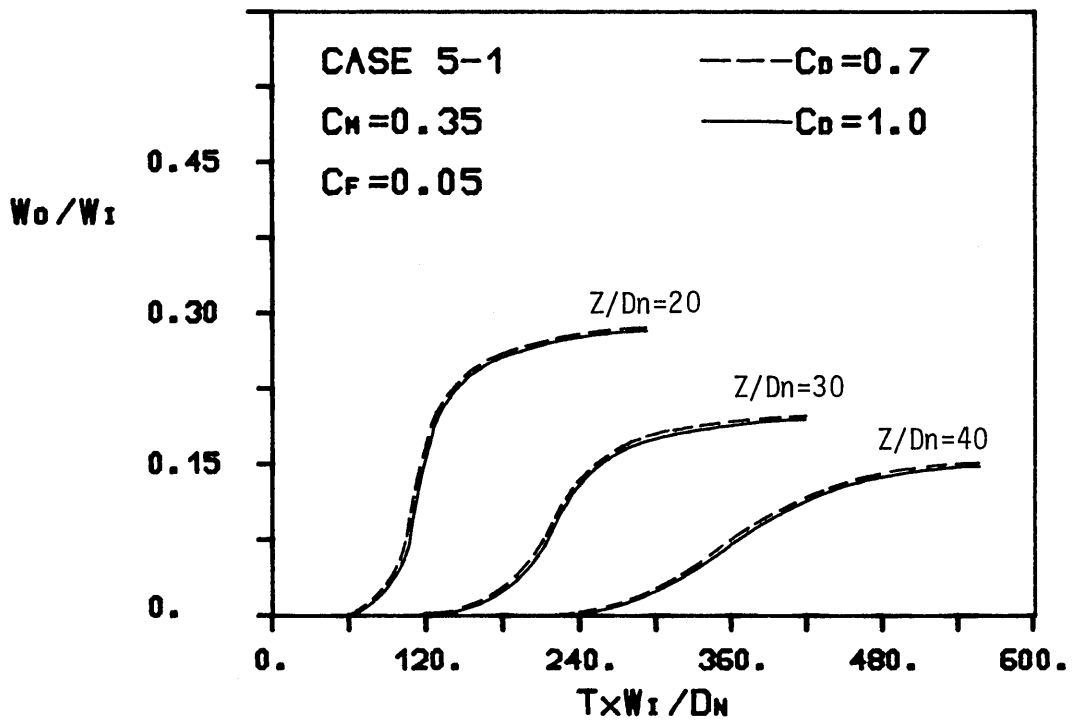


FIG. 5.5-10 EFFECT OF C_D ON MEAN VELOCITY AT GIVEN AXIAL STATIONS

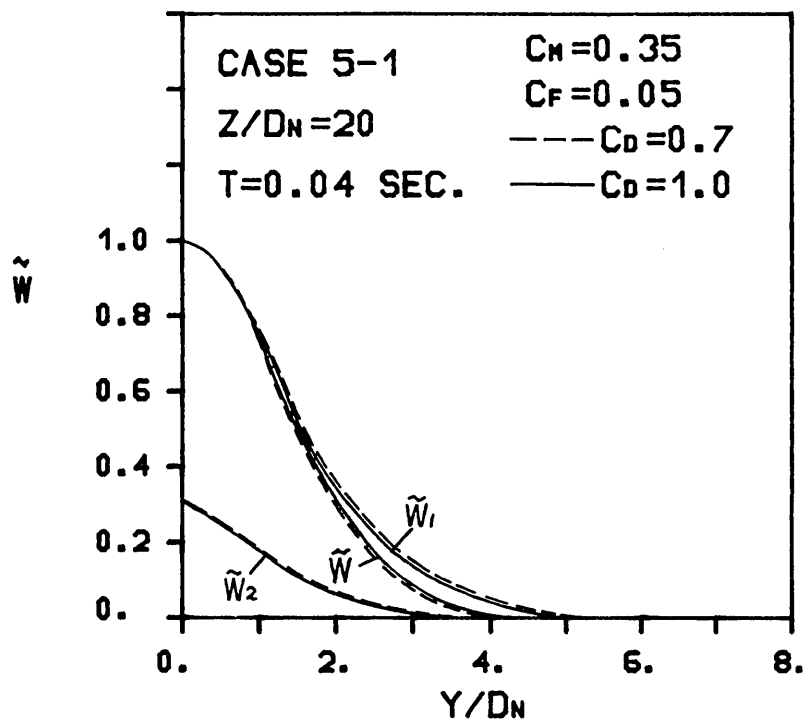


FIG. 5.5-11 EFFECT OF C_D ON THE PREDICTED PROFILES OF STREAMWISE VELOCITIES

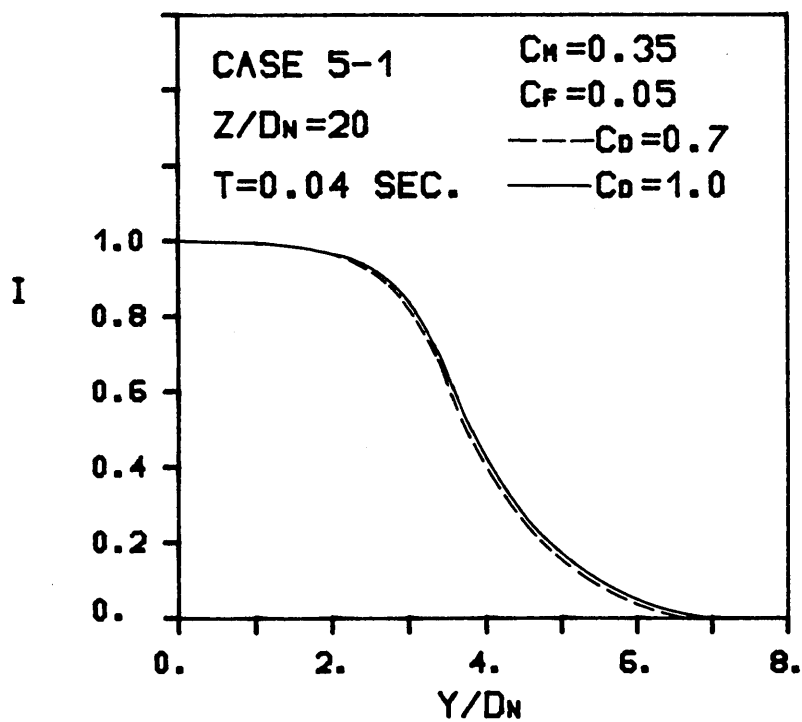


FIG. 5.5-12 EFFECT OF C_D ON THE PREDICTED PROFILES OF INTERMITTENCY FACTOR

pressure conditions. The different distances from the axis to the outer boundary of the domains may cause the discrepancy which has just been mentioned.

Figs. 5.4-2, 5.4-3, and 5.4-4 show the predicted and measured developments of the mean velocity at given axial stations for the three cases considered. It can be seen, that the predicted time-delays for the tip of the transient jets travelling from the nozzle exit to the axial stations agree with the experimental data quite well for all the stations considered. It has also been found, that the mean velocity of the transient jet attains that of the steady jet. The steady jet was considered in Chapter 4.

But the predicted rises in the mean velocities at these stations are much more gradual than the data, resulting in large steady-state attainment time. One of the reasons for it may be the modelling assumptions made in the computations. In this work, the turbulent viscosity was given by a constant-effective-viscosity model which assumes the viscosity as invariant with time. In reality, it would be small initially, and then increases. Therefore, the constant-effective-viscosity assumption overestimates the phenomenon at first, and underestimates it later.

In addition, the mixing length was taken to be the mean

fragment size L . It can be found from equation (2.42-4), that the mixing length depends on only the axial distance from the nozzle exit, and does not depend on time. Spalding (1982a) suggested that the mean fragment size can be deduced from a separate transport equation. The turbulent viscosity as well as the transport laws may then be obtained by using the fragment size varying with time. Using this method, an improved prediction of the transient flow can be expected.

On the other hand, any small errors in the experiment resulting from an inaccurate assessment of the instruments' sensitivity and calibration could be significant in the measurement of this kind where the time-scale involved was of the order of milli-seconds.

Hengrussamee (1975) predicted the same transient jet by using the $K - \epsilon$ or constant-effective-viscosity models. The constants for the $K - \epsilon$ model were those which Launder and Spalding (1973) found appropriate to plane jets and mixing layers. Using the $K - \epsilon$ model, the predicted velocity rises at the axial stations were in a good agreement with the experimental data; but the time-delays for the tip of the transient jet travelling from the nozzle exit to the stations were about half of the data. With constant-effective-viscosity model, Hengrussamee's computations had similar trends to those of the present

calculation; but his predicted velocity rises agreed with the data better than this work. However, the effect of the grid-node number on his predictions was not reported in detail. He used 22×22 grid-nodes which, in this work, have proved to be too coarse (see Figs. 5.32-1 and 5.32-2).

Figs. 5.5-1 to 5.5-12 show the effects of the constants C_m , C_f , and C_d of the two-fluid assumptions on the predictions. As in Chapter 4, it has been found that all the changes in the constants did not exert considerable influence on the predictions of the mean flow.

Fig. 5.5-4 shows, that an increase in C_m leads to an increase in the intermittency factor I , as more non-turbulent fluid is entrained by the turbulent one.

Fig. 5.5-7 shows that an increase in C_f leads to a decrease in the slip between the two fluids as expected.

5.7 Conclusion

The air-into-air transient jet has been simulated by employing the mathematical model with the constant-effective-viscosity and the two-fluid assumptions. Predictions of the mean velocity along the centre-line, the developments of the streamwise velocity of the mean flow, the profiles of the streamwise velocities and

intermittency factor at given axial stations have been obtained. The computational results have been compared with those of the steady jet and available experimental data.

The performance of the mathematical model for this transient flow can be summarized as follows:

- (1) With time, the results obtained for the transient jet approach those of the steady jet.
- (2) Agreement of the prediction with the experimental data of the time-delay for the tip of the transient jet travelling from the nozzle exit to the given axial stations is satisfactory; and
- (3) The discrepancy of the time for the transient jet to reach the steady-state is found between the prediction and the experimental data.

The modelling assumptions with the empirical constants presented in Chapter 2 can be used for the flow considered in this chapter; but refinements of these assumptions are required by allowing the mean fragment size as well as the turbulent viscosity to vary with time, in order to improve the transient calculation. This remains a topic for future work.

CHAPTER 6 - EXPERIMENTAL INVESTIGATION

6.1 Introduction

The previous two chapters presented the numerical simulations of air-into-air steady and transient jets. The computational results were compared with existing experimental data. In this chapter, an experimental investigation is presented. The purposes of this work may be summarized as follows:

- (1) To observe the similarities between miscible (brine-into-water) and immiscible (kerosene-into-water) transient jets.
- (2) To measure the tip-penetration rates of the transient jets in (1) above.
- (3) To measure the droplet size of a kerosene-into-water turbulent jet under steady-state conditons; and
- (4) To measure the time-dependent boundary conditions at the nozzle exit of the two transient jets.

An experimental rig has been constructed in the present study. It can be used for performing both steady and transient jet experiments, as well as for measuring the tip-penetration rate and the development of the volume

flow rate of the injected fluids of the transient jets.

For the experiment of the miscible transient jet, brine was injected vertically downwards into tap water which was initially stagnant forming a single-phase flow. For the immiscible transient jet, kerosene was injected vertically upwards into the water forming a two-phase flow.

For the transient jet, the volume flow rate of the injected brine or kerosene was developing with time. The volume flow rate was measured and a Spline Function was then fitted through the data. This function will be used as a boundary condition in the jet simulations.

For the kerosene-into-water steady jet, the droplet size of the fragmented kerosene has been measured using a photographic technique. The data of the mean droplet diameter will be used in the computations; this will be described in Chapter 8.

The remainder of this chapter consists of five sections in which Section 6.2 describes the apparatus, the experimental procedure is described in Section 6.3, the experimental results are reported in Section 6.4. Section 6.5 is devoted to the discussion, followed by the closure in Section 6.6.

6.2 Apparatus

The apparatus is now described as follows:

(1) General arrangement

A schematic representation and an overall view of the experimental apparatus are provided in Figs. 6.2-1 and 6.2-2 respectively.

It can be seen from Fig. 6.2-1, that there is a main tank 5 and a glass jar 1 as the containers for the liquids used in the experiments. The dimensions of the main tank are 0.4m (length) x 0.4 m (width) x 0.6 m (height), and this is shown in detail in Fig. 6.2-3. The bottom of the tank is a perspex plate 0.5 in. thick while the walls of the tank are made from 10 mm thick glass. A valve 11 is used to drain the tank.

For the kerosene-into-water experiment, a 0.25 in. inside diameter perspex tube 9 was fixed at the centre of the tank bottom. The lower end of the tube was connected to a flowmeter 12 via a polythene tubing. The nozzles with various inside diameters used in different experiments, were inserted into the perspex tube one by one.

For the brine-into-water experiment, a perspex tube 8 which is identical to the perspex tube 9, was fixed onto a

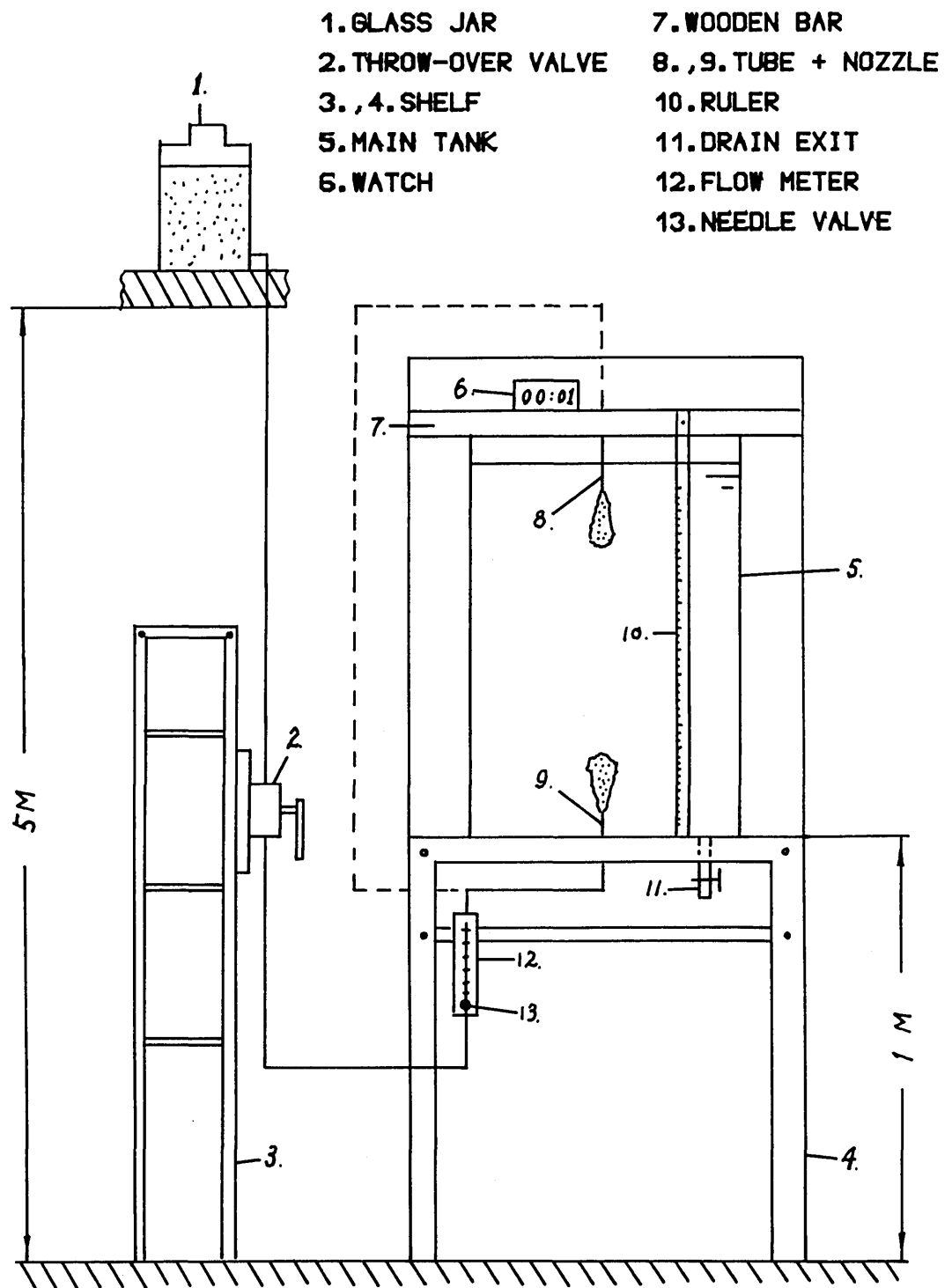


FIG. 6.2-1 SCHEMATIC REPRESENTATION OF THE GENERAL LAY-OUT OF THE EXPERIMENTAL APPARATUS

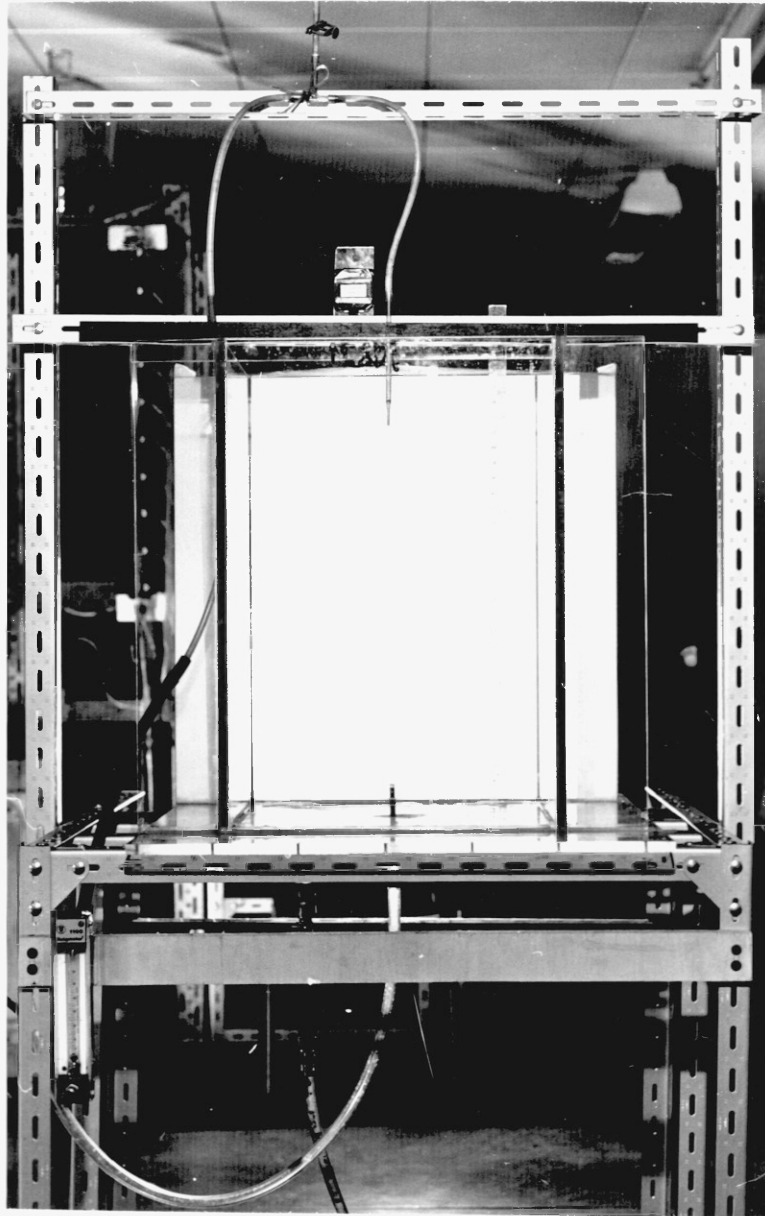


Fig. 6.2-2 Overall view of apparatus

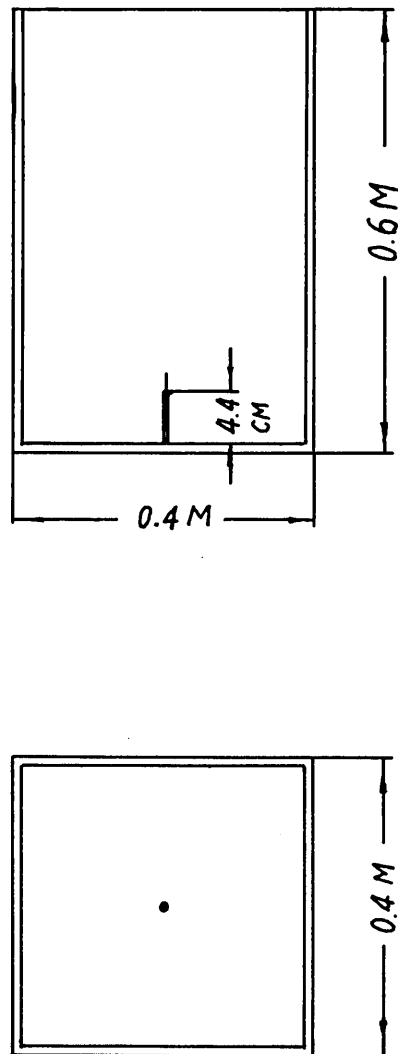


FIG. 6.2-3 DIMENSIONS OF THE MAIN TANK

wooden bar 7, and placed at the centre of the top plane of the tank. The nozzles were inserted into the tube which was connected to the flowmeter.

The glass jar 1 is placed on the laboratory gallery about 5 m above the floor, this provides pressure head required to drive the liquid from the jar into the tank.

A throw-over valve 2 and a flowmeter 12 together with a needle valve 13 are placed between the glass jar and the main tank. The throw-over valve and the flowmeter are fixed onto the shelves 3 and 4 respectively.

During the experiment, a digital watch 6 and a ruler 10 were mounted onto the wooden bar, to enable the time and scale to be measured from the photographic negatives. There has not been synchronous device to simultaneously open the valve and start the watch.

(2) Valve system

The valve system consists of a throw-over valve 2 and a needle valve 13. The volume flow rate of the injected fluid under a steady-state discharge condition, is controlled by the sensitive needle valve with the throw-over valve open. For the transient jet, the volume flow rate is first preset under a steady-state condition. The throw-over valve is then closed leaving the needle valve

open as before. After rapidly opening the throw-over valve again the motion of the jet starts from rest. It can be, therefore, ensured that the volume flow rate of the injected fluid, when the transient jet eventually reaches a steady-state, is equal to the steady-state preset value.

(3) The nozzles

Three stainless steel tubes with smooth walls and exits were used as jet nozzles. Their inside diameters are 1.6, 2.0, 2.6 mm respectively.

(4) Instruments

- Flowmeter

A "Fisher Controls" Series 1100 flowmeter was used. Its range is from 30 cc/min to 50 l/min for air at 15° c and 760 mm Hg Abs, or from 1 cc/min to 1400 cc/min for water at 20° c. Since the scale reading on the flowmeter was calibrated for air and water, the volume flow rate for the kerosene and brine in this work had to be obtained from the calibration charts supplied by the manufacturer.

- Hydrometer

A set of "Tower Streamline Specific Gravity Hydrometer"

was used.

- Photographic instruments

The camera set used to take the photographs was a Cannon A1 fitted with a 1:1.8 lens and a Cannon MA SET motor drive with the highest winding speed at 5 frames per second. In order to improve the contrast between the flow patterns and the background, the main tank was illuminated from behind by a 1000-watt IQ photographic lamp with the light passing through a diffuser screen. For a film rated at 400 ASA the exposure was generally set to 1/1000 sec. at f 2.0.

- Video recorder

The developments with time of the volume flow rate of the injected fluids were recorded by a "SONY HVS-4000 P" video camera and a "SONY SL-C9UB PAL" video cassette recorder.

- Library reader

The tip-penetration rates and the droplet sizes were measured from the photographic negatives taken in the experiments. These negatives were magnified by a "Recordak Model A.H.3" Library Reader having two lenses with "24 x" and "12 x" magnifications.

6.3 Experimental procedure

All experimental procedures may be divided into three stages which are now described.

(1) Preparing the experiment

- Decide the experimental parameters, such as nozzle diameter, liquid to be injected, volume flow rate of the injected fluid, etc.;
- Clean the main tank from all wastes left from the previous run;
- Insert the nozzle with the predetermined diameter into the perspex tube; and
- Level the main tank.

(2) Preparing the injected liquids

For the brine-into-water transient jet experiment, the injected brine was made by dissolving salt in water. Both the injected kerosene and brine were dyed. Waxolin was used for dyeing the kerosene while Nigrosine was for the brine.

(3) Operation of the experiment

Once all the procedures described in (1) and (2) above were finished, the experiment was ready to be run. The operation of the experiment can be described as follows:

- Open the throw-over valve, then adjust the needle valve under a steady-state condition, until the scale reading of the flowmeter reaches the predetermined value;
- Close the throw-over valve leaving the needle valve open as before;
- For the brine-into-water experiment, drain off the used water from the main tank, and then fill again with clean tap water;
- For the kerosene-into-water experiment, collect the kerosene floating on the water;
- Switch on the watch;
- In the transient jet experiments, simultaneously open the throw-over valve and switch on the motor drive of the camera to take photographs. Typically, for a transient jet, 5-10 frames could be taken for each run covering the period during which the jet tip travelled

about 150 nozzle diameters;

- For the droplet-size measurement, take the photograph for the kerosene-into-water jet under a steady-state condition;
- Repeat the operation for the transient jet experiment described above, to record the flowmeter reading using the video camera and video cassette recorder.

6.4 Experimental results

6.41 Experimental conditions

Table 6.41-1 gives the experimental conditions for the brine-into-water and kerosene-into-water transient jets. Table 6.41-2 gives those for the droplet-size measurement of the kerosene-into-water steady jet, while Table 6.41-3 gives the physical properties of the liquids used in the experiments.

In the tables,

- D_n is the diameter of the nozzle;
- V_i and W_i are the volume flow rate and velocity of the injected fluids at the nozzle exit. For the case of the transient jet, V_i and W_i are then the values when

Case	Fluids	Dn (mm)	Vi (10^{-6} m ³ /sec.)	Wi (m/sec.)	Re
6-1	Brine- into- water	1.6	4.21	2.09	3,344
6-2		1.6	8.42	4.18	6,688
6-3		2.0	4.21	1.34	2,680
6-4		2.6	8.42	1.59	4,108
6-5	Kerosene- into- water	1.6	3.77	1.88	1,527
6-6		1.6	5.97	2.88	2,339
6-7		2.0	6.57	2.09	2,122
6-8		2.6	11.01	2.07	2,732

Table 6.41-1 Experimental conditions of transient jet

Case	Fluids	Dn (mm)	Vi (10^{-6} m ³ /sec.)	Wi (m/sec.)	Re
6-9	Kerosene- into- water	1.6	2.33	1.16	942
6-10		1.6	3.77	1.88	1,527
6-11		1.6	5.79	2.88	2,339
6-12		2.0	3.43	1.09	1,106
6-13		2.0	5.26	1.68	1,695
6-14		2.0	6.57	2.09	2,122
6-15		2.6	7.18	1.35	1,769
6-16		2.6	9.08	1.71	2,259
6-17		2.6	11.01	2.07	2,732

Table 6.41-2 Experimental conditions of droplet-size measurement

	Density (kg/m ³)	Viscosity (Ns/m)	Interfacial tension (mN/m)
Water	998	1.002×10^{-3}	-
Brine	1,185	-	-
Kerosene	785	1.55×10^{-3}	48

Table 6.41-3 Physical properties of the
experimental liquids of 20⁰c

Source: Density: measured

Others: Rose and Cooper (1977)

the jet reaches a steady-state;

- Re is the Reynolds number which was defined in equation (4.2-1);

6.42 Results of brine-into-water and kerosene-into-water transient jets

(1) Typical flow patterns

Fig. 6.42-1 shows typical flow patterns of a miscible transient jet (brine-into-water, case 6-2) and an immiscible transient jet (kerosene-into-water, case 6-6). The successive photographs which show their growths with time are arranged according to the dimensionless variable $TxWi/Dn$.

(2) Tip-penetration rates

The tip-penetration rates of the transient jet were obtained from a visual examination of the magnified photographic negatives taken from the experiments. Data from 2, 3 or 4 runs were provided for each case considered. These data are presented in Figs. 6.42-2 to 6.42-5 for the brine-into-water transient jet and in Figs. 6.42-6 to 6.42-9 for the kerosene-into-water transient jet. In these figures, dimensionless variables Ht/Dn and $TxWi/Dn$ are adopted to be the measures of height of the jet-tip and elapsed time.

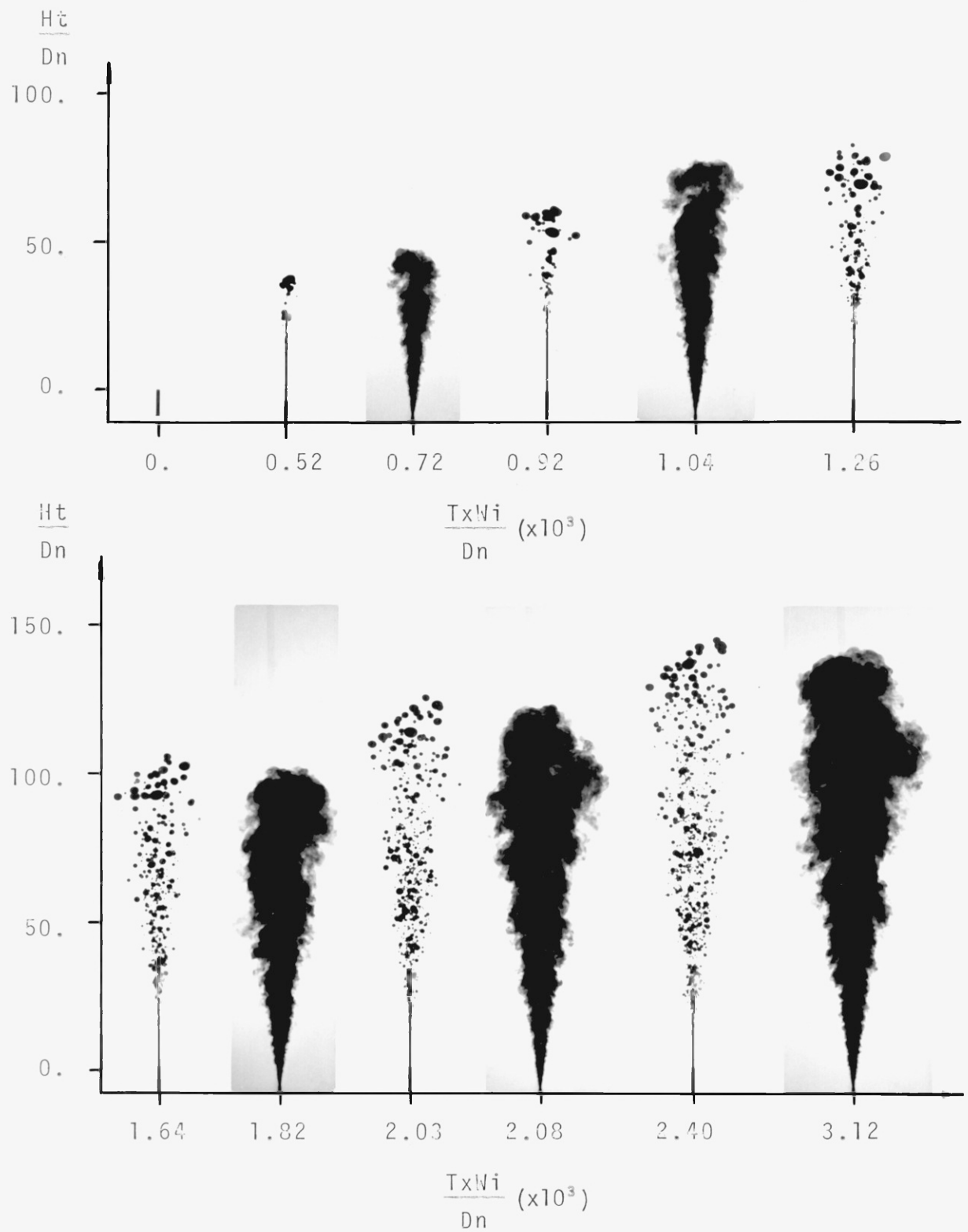


Fig. 6.42-1 Flow patterns of a brine-into-water and a kerosene-into-water transient jets

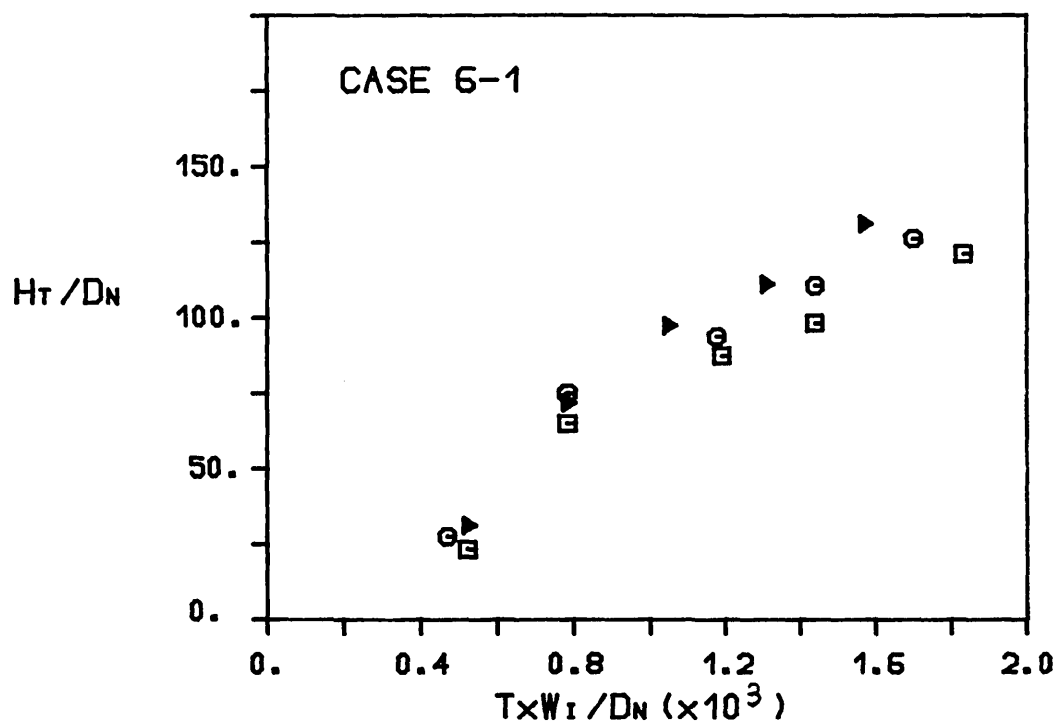


FIG. 6.42-2 EXPERIMENTAL DATA OF THE TIP-PENETRATION OF A BRINE-INTO-WATER TRANSIENT JET: CASE 6-1

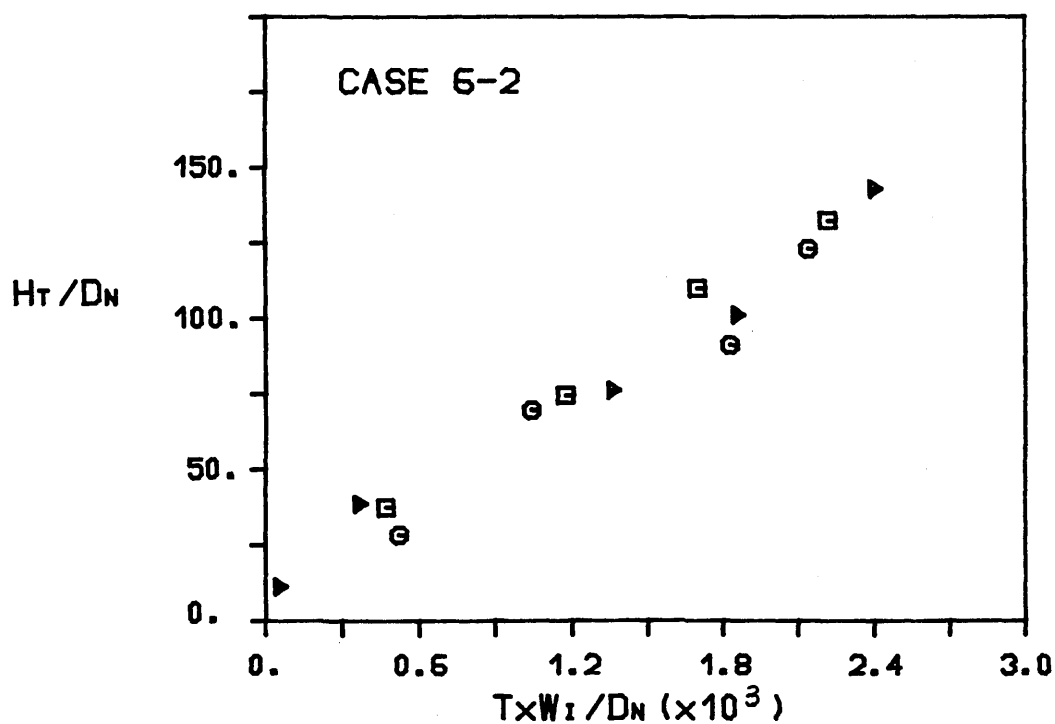


FIG. 6.42-3 EXPERIMENTAL DATA OF THE TIP-PENETRATION OF A BRINE-INTO-WATER TRANSIENT JET: CASE 6-2

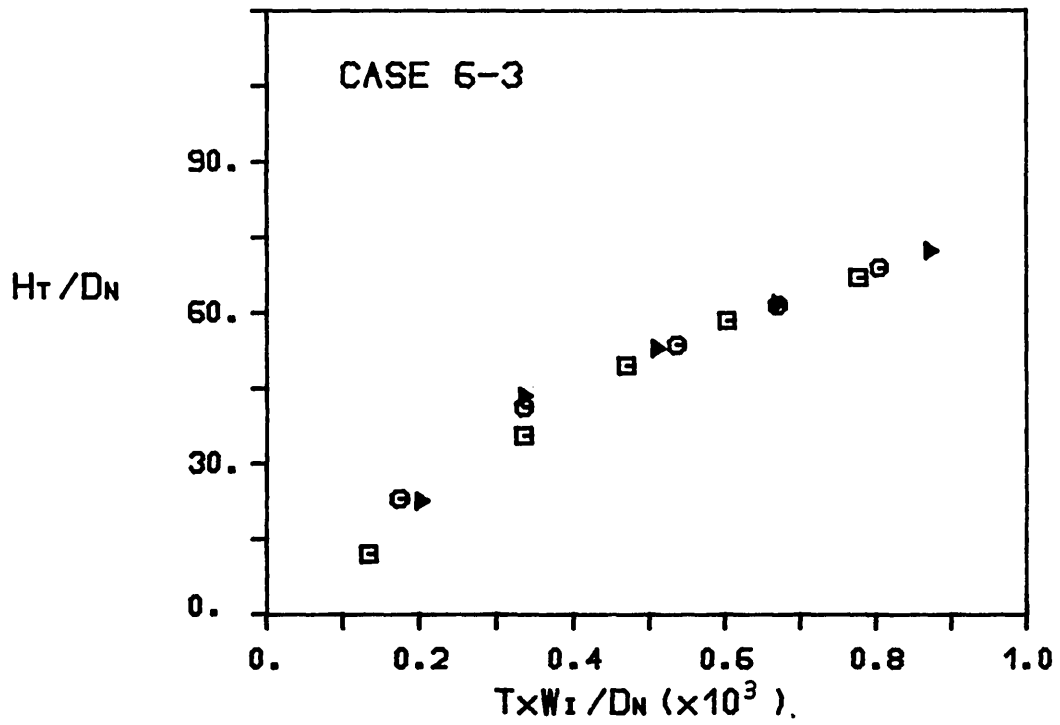


FIG. 6.42-4 EXPERIMENTAL DATA OF THE TIP-PENETRATION OF A BRINE-INTO-WATER TRANSIENT JET: CASE 6-3

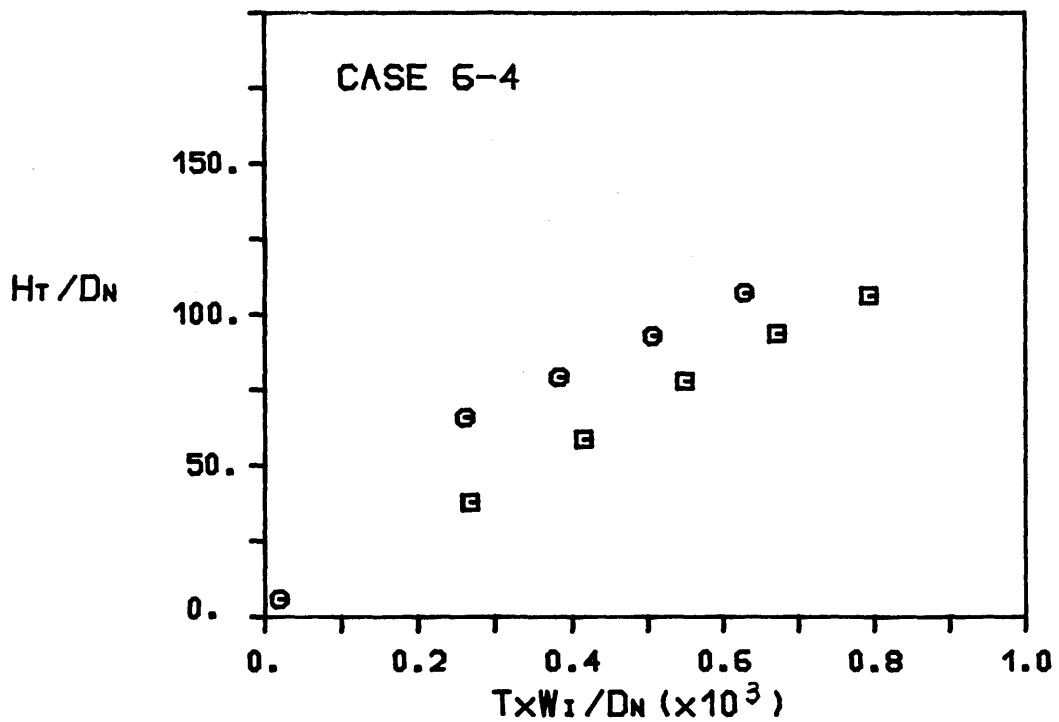


FIG. 6.42-5 EXPERIMENTAL DATA OF THE TIP-PENETRATION OF A BRINE-INTO-WATER TRANSIENT JET: CASE 6-4

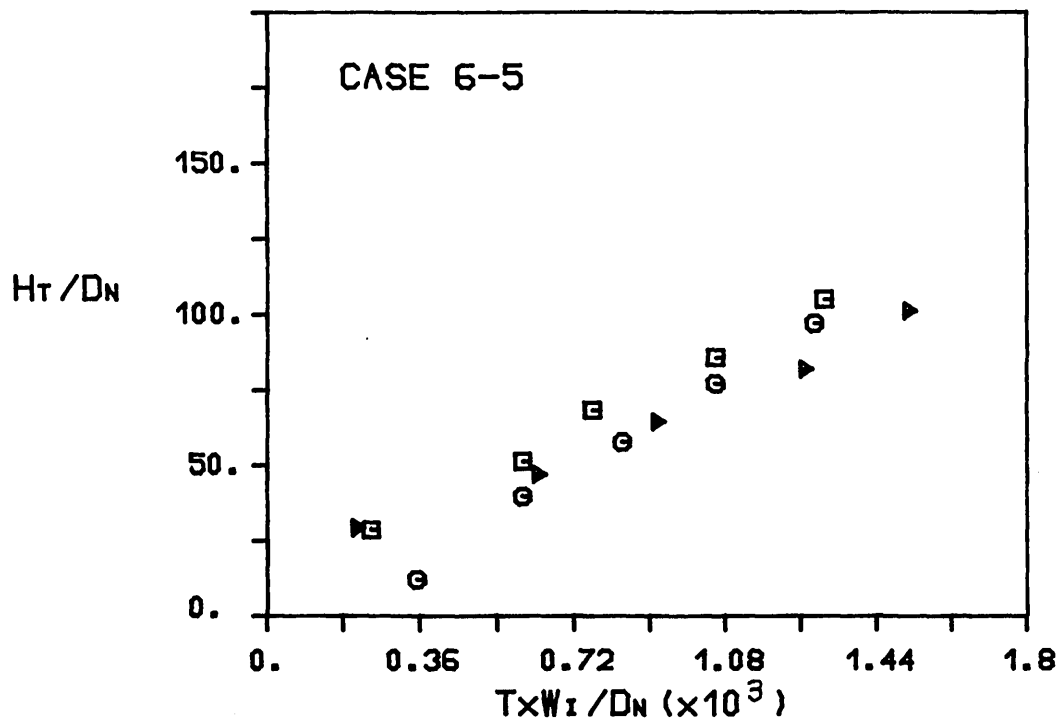


FIG. 6.42-6 EXPERIMENTAL DATA OF THE TIP-PENETRATION OF A KEROSENE-INTO-WATER TRANSIENT JET: CASE 6-5

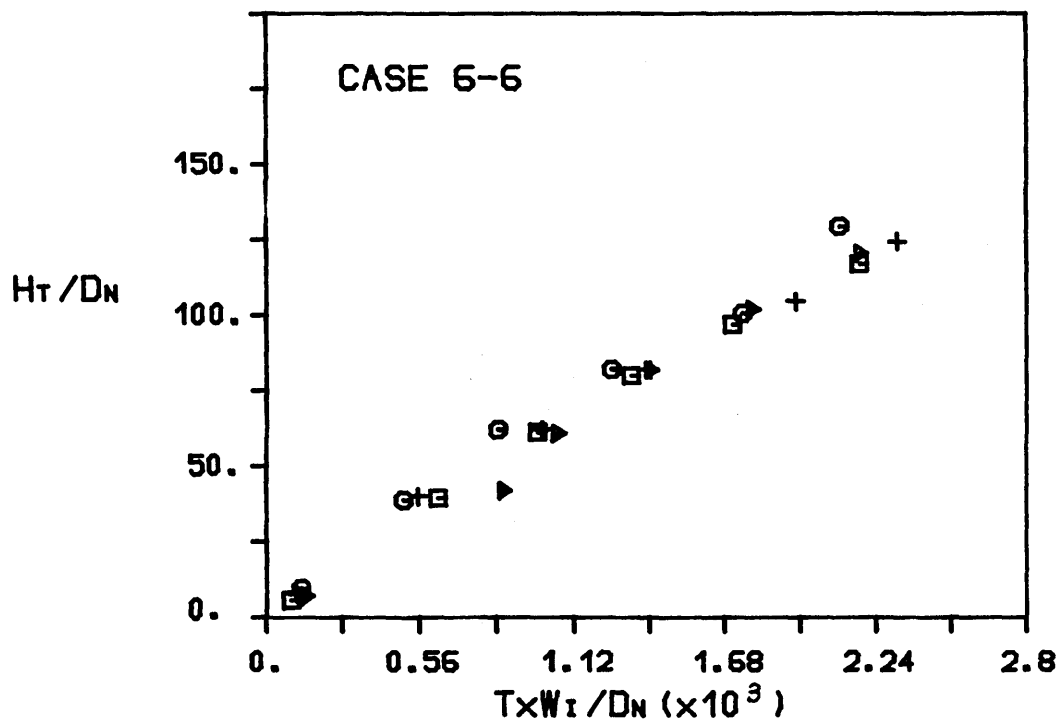


FIG. 6.42-7 EXPERIMENTAL DATA OF THE TIP-PENETRATION OF A KEROSENE-INTO-WATER TRANSIENT JET: CASE 6-6

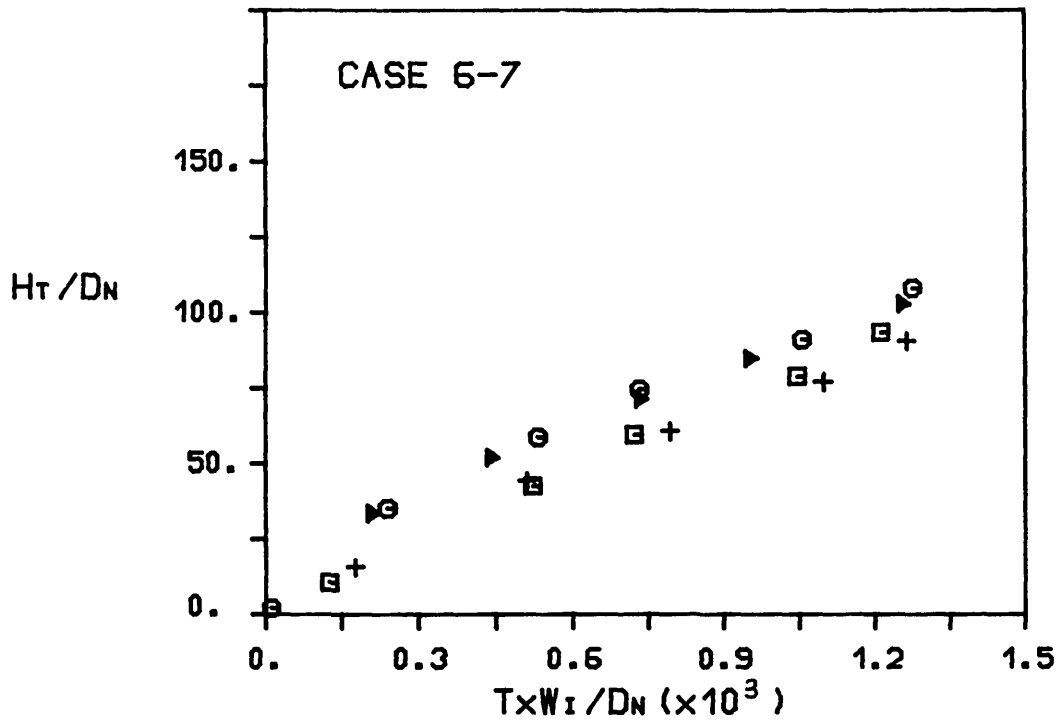


FIG. 6.42-8 EXPERIMENTAL DATA OF THE TIP-PENETRATION OF A KEROSENE-INTO-WATER TRANSIENT JET: CASE 6-7

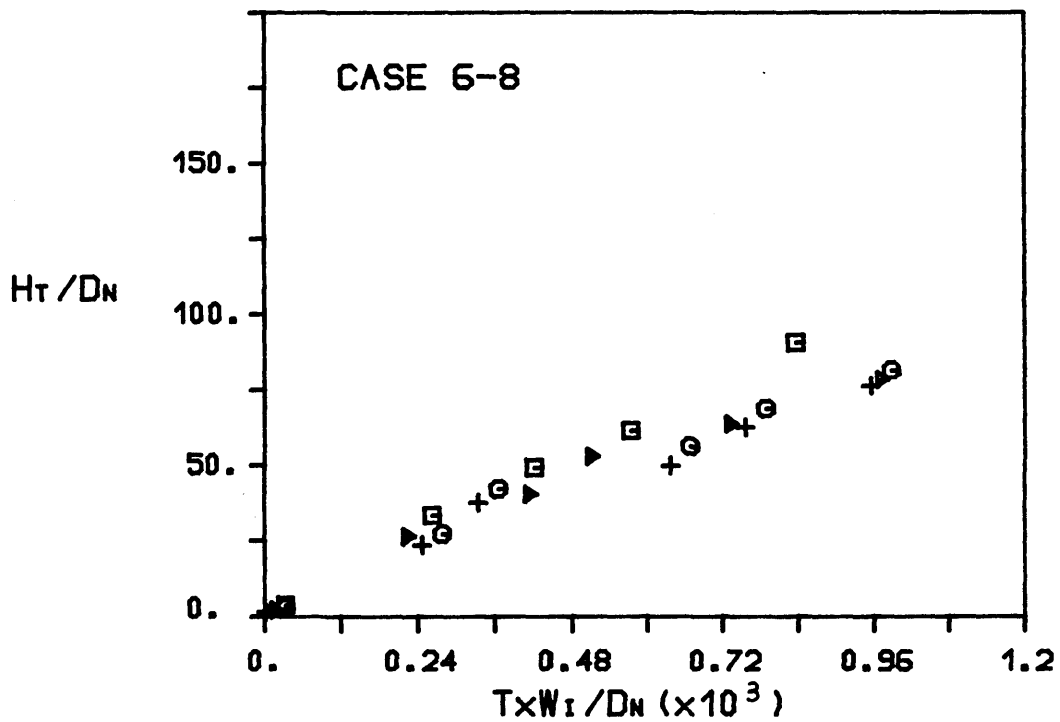


FIG. 6.42-9 EXPERIMENTAL DATA OF THE TIP-PENETRATION OF A KEROSENE-INTO-WATER TRANSIENT JET: CASE 6-8

6.43 Volume flow rate of injected fluids

The volume flow rate of the injected brine or kerosene was obtained as follows:

- Record the variation with time of the ball position of the flowmeter on a video tape;
- Replay the video tape frame by frame and read the ball position for each frame;
- 25 frames per second is the normal running speed of the recorder, the time between two successive frames can then be known;
- Deduce the volume flow rate of the injected fluids from the ball positions using the calibration charts supplied by the manufacturer.

A Spline Function is applied in the smoothing of the experimental data. It is supposed, that to each t_i there corresponds an equally erratic measurement V_i . The following Spline Function is defined for above volume flow rate versus time data:

$$V_i(t) = A_i + B_i(t-t_i) + C_i(t-t_i)^2 + D_i(t-t_i)^3 \quad (6.43-1)$$

$$t_i \leq t < t_{i+1} \quad i = 1, 2, 3, \dots, \text{ and}$$

V, V', V'' are continuous.

where $V_i(t)$ is the volume flow rate as a function of time t , and the coefficients A_i, B_i, C_i , and D_i were determined using the program "Spline". Its listing is given in Appendix B. The Spline Function is explained in detail by Handscomb (1965).

The experimental data and the fitted Spline Functions are presented in Figs. 6.43-1 to 6.43-6 for all the transient cases considered.

6.44 Results of droplet-size measurement of kerosene-into-water steady jet

(1) Overall views of fragmented jets

Figs. 6.44-1, 2, and 3 show typical overall views of the fragmented kerosene-into-water steady jets for the steady cases considered.

(2) Typical volume percentage of droplets in diameter-size groups in a cross-section

Volume percentage of droplets in diameter-size groups in a cross-section was obtained from the magnified negatives as

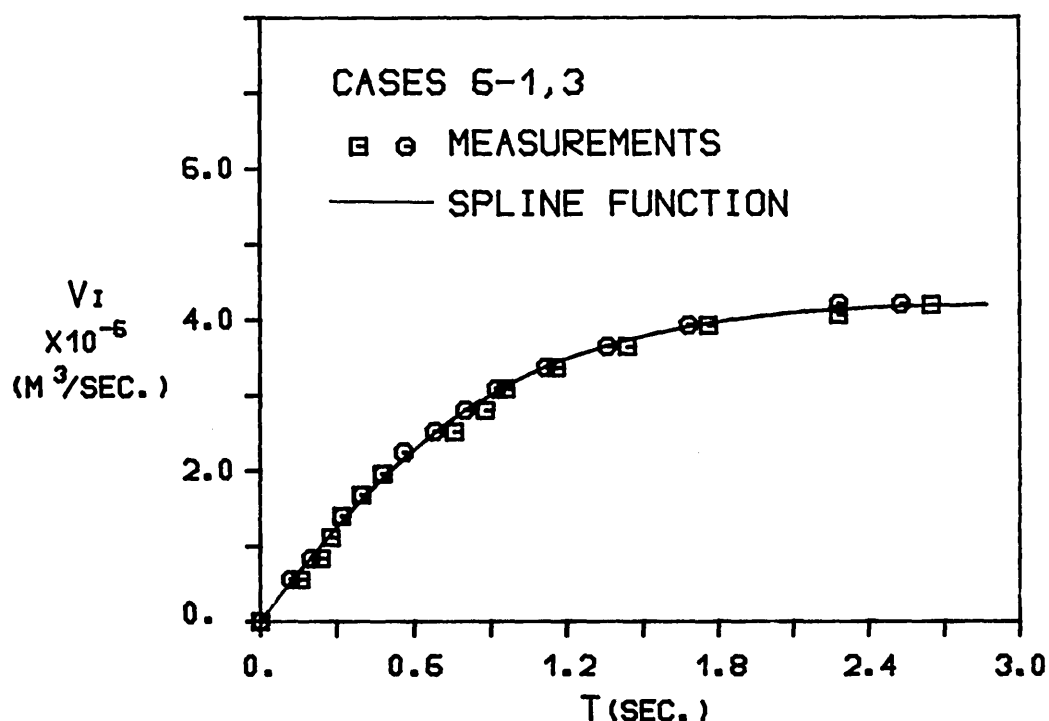


FIG. 6.43-1 EXPERIMENTAL DATA OF THE VOLUME FLOW RATE AT THE EXIT OF BRINE-INTO-WATER TRANSIENT JETS: CASES 6-1,3

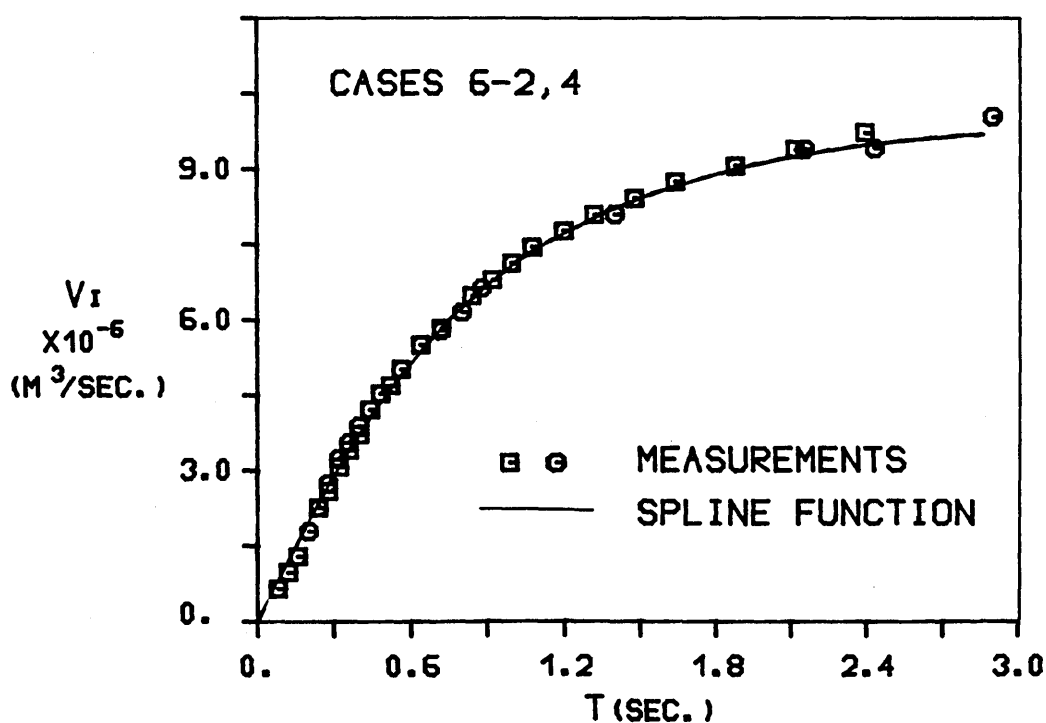


FIG. 6.43-2 EXPERIMENTAL DATA OF THE VOLUME FLOW RATE AT THE EXIT OF BRINE-INTO-WATER TRANSIENT JETS: CASES 6-2,4

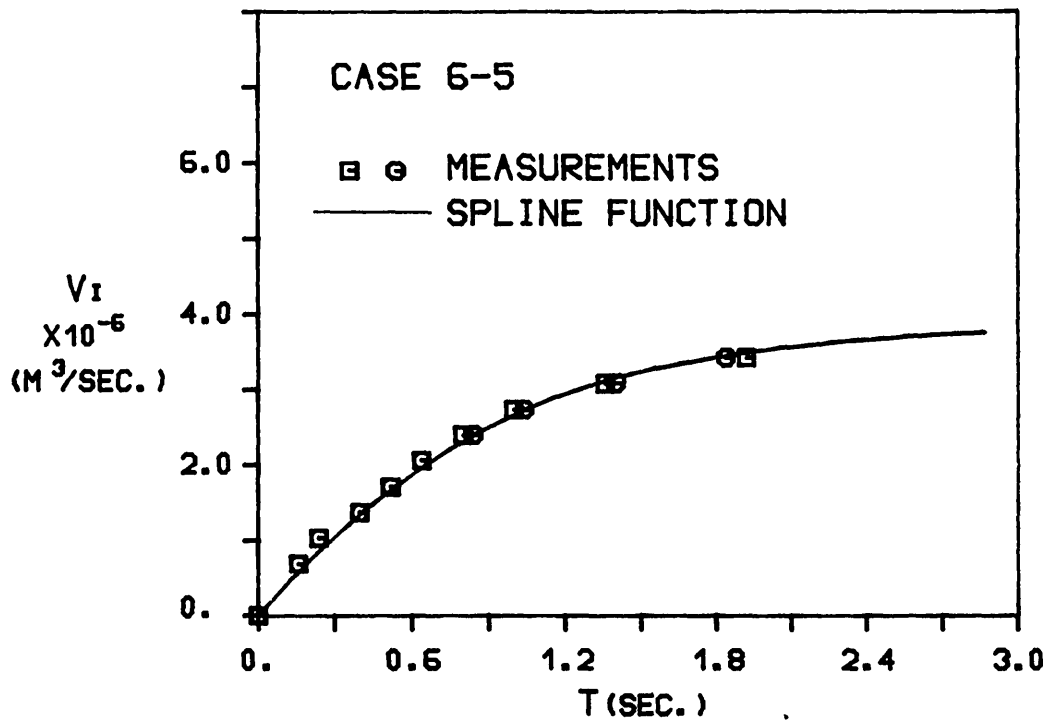


FIG. 6.43-3 EXPERIMENTAL DATA OF THE VOLUME FLOW RATE AT THE EXIT OF A KEROSENE-INTO-WATER TRANSIENT JET: CASE 6-5

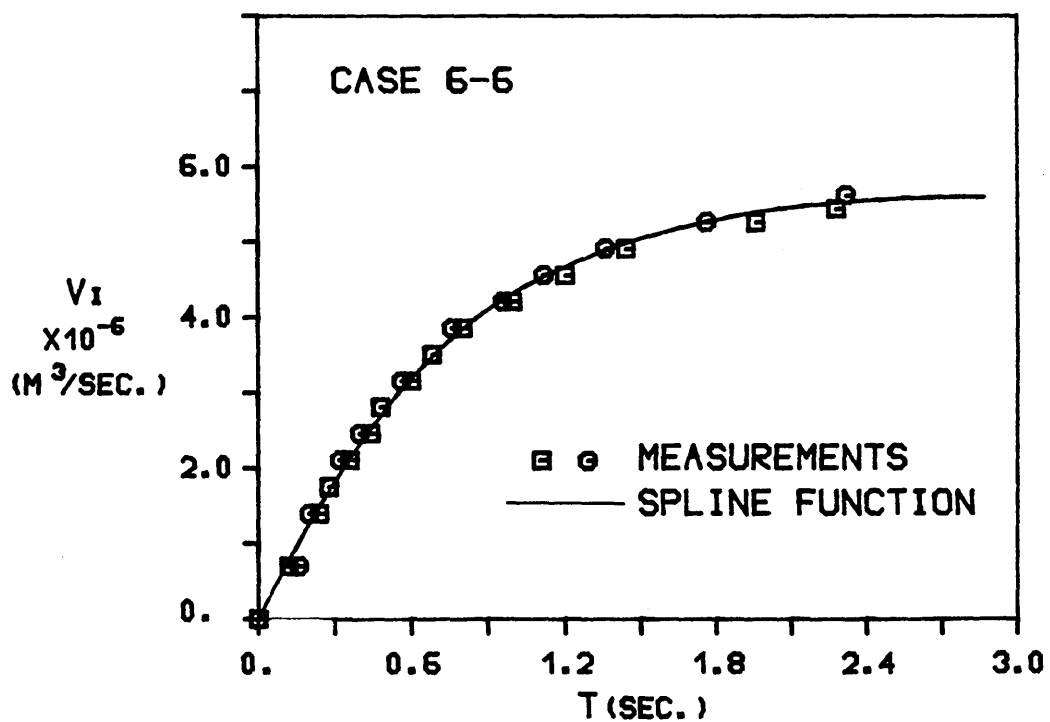


FIG. 6.43-4 EXPERIMENTAL DATA OF THE VOLUME FLOW RATE AT THE EXIT OF A KEROSENE-INTO-WATER TRANSIENT JET: CASE 6-6

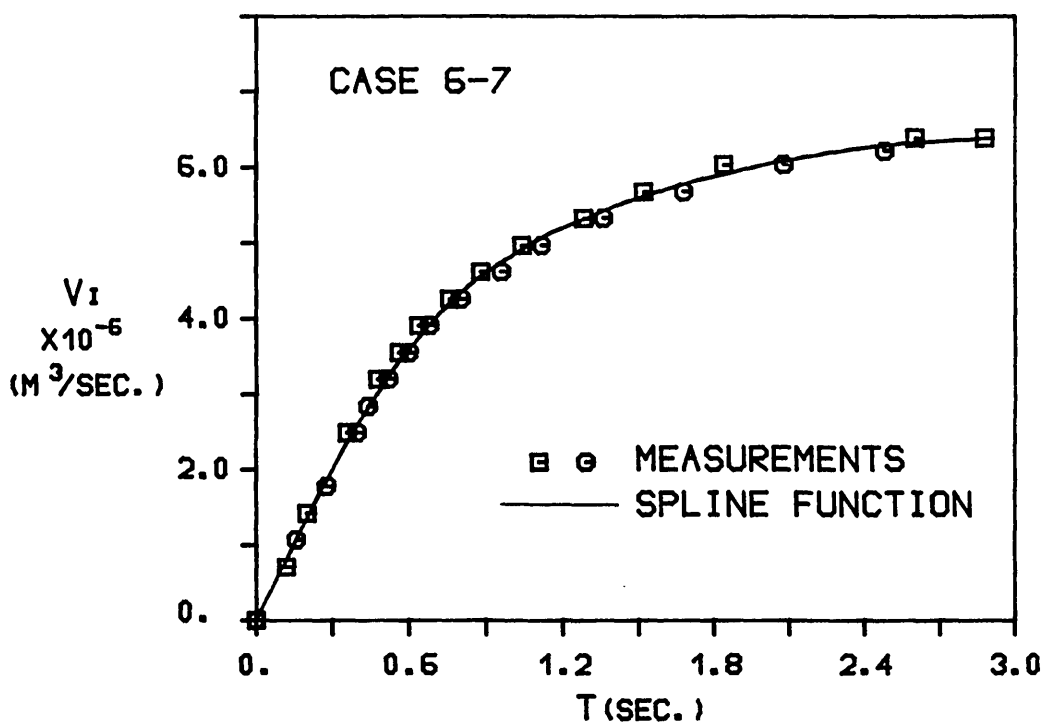


FIG. 6.43-5 EXPERIMENTAL DATA OF THE VOLUME FLOW RATE AT THE EXIT OF A KEROSENE-INTO-WATER TRANSIENT JET: CASE 6-7

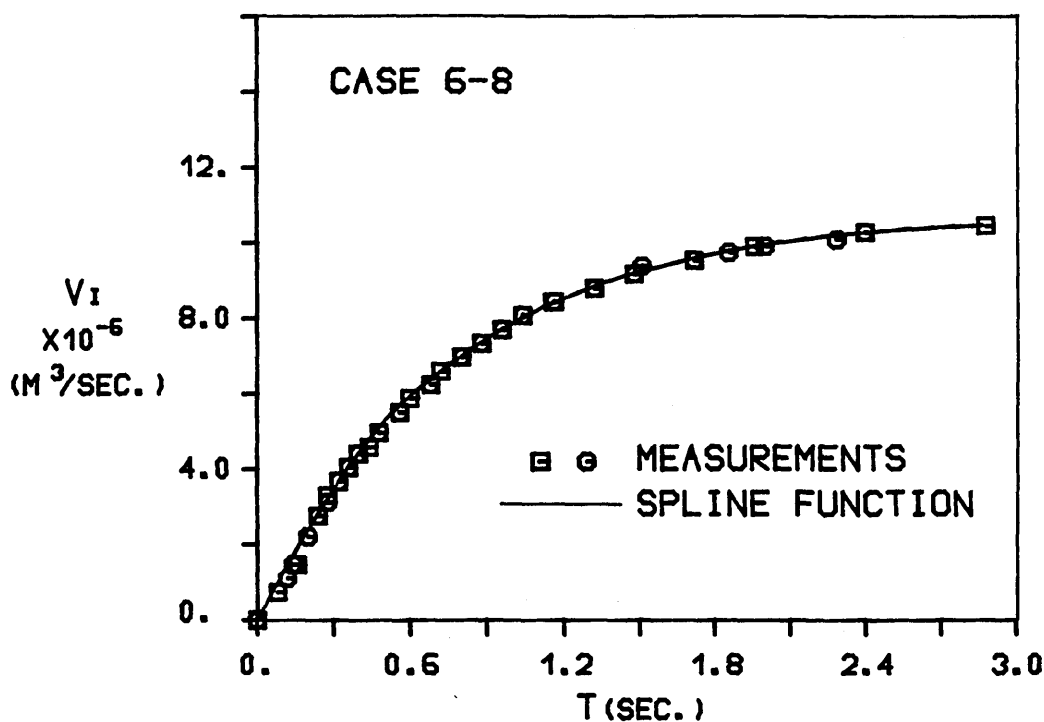


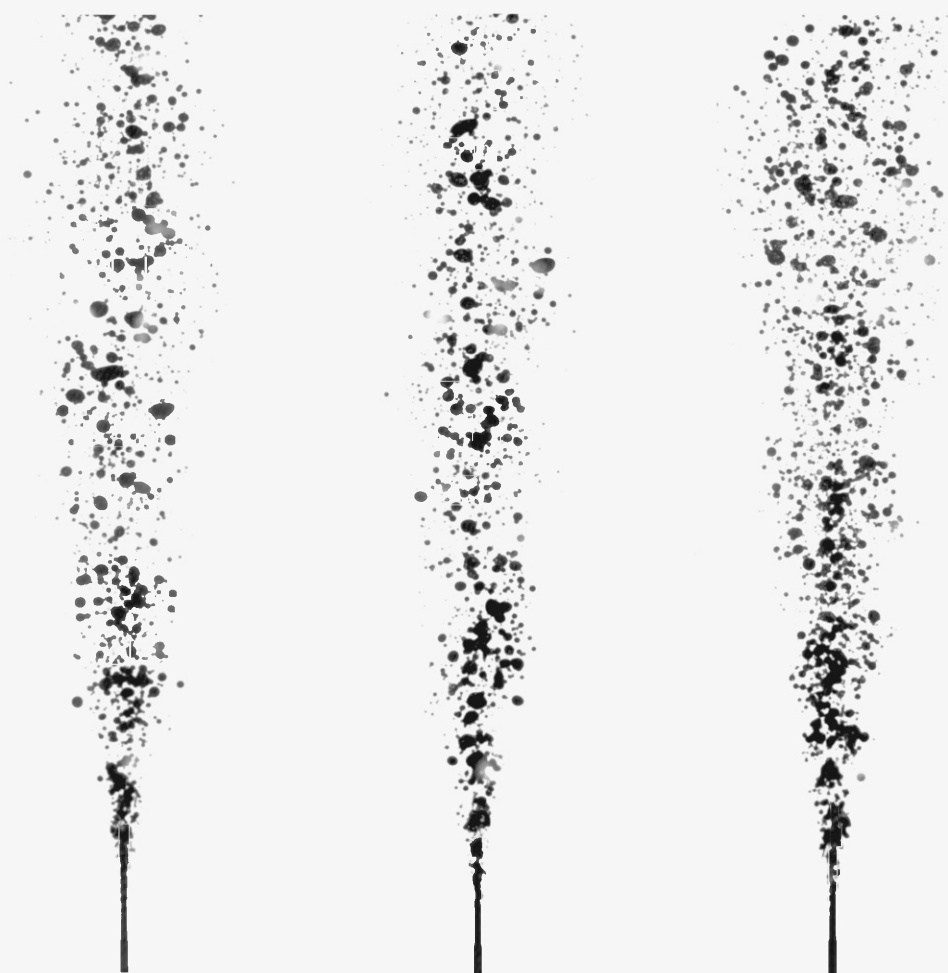
FIG. 6.43-6 EXPERIMENTAL DATA OF THE VOLUME FLOW RATE AT THE EXIT OF A KEROSENE-INTO-WATER TRANSIENT JET: CASE 6-8



Fig. 6.44-1 Overall views of kerosene-into-water
steady jet: Cases 6-9, 6-10, 6-11



Fig. 6.44-2 Overall views of kerosene-into-water
steady jet: Cases 6-12, 13, 14



Case 6-15

Case 6-16

Case 6-17

Fig. 6.44-3 Overall views of kerosene-into-water
steady jet: Cases 6-15, 16, 17

follows:

- Magnify a negative on the screen of the library reader with a transparent scaled sheet attached to the screen;
- Take a cross-section from the flow region;
- Divide the diameter of the droplets in this section into a number of diameter-size groups;
- Count the number of the droplets falling into each size group;
- Most of the droplets were circular, only a few of them were elliptic. For these, the diameter was taken as half the sum of their long and short axes;
- Measure 24-30 negatives for each case;
- Calculate the volume of the droplets in each group;
- Calculate the total volume of all droplets within this cross-section;

The volume percentage of the droplets in a cross-section was thus obtained.

Figs. 6.44-4 (1), (2), and (3) show the volume percentages of the droplets within a cross-section $Z=50D_n-60D_n$ for all the cases considered. In these figures, V_d is the volume percentage of the droplets, d is the diameter of the size groups taken, while d_c is the "characteristic diameter" which is given by Ishii and Chawla (1979) as:

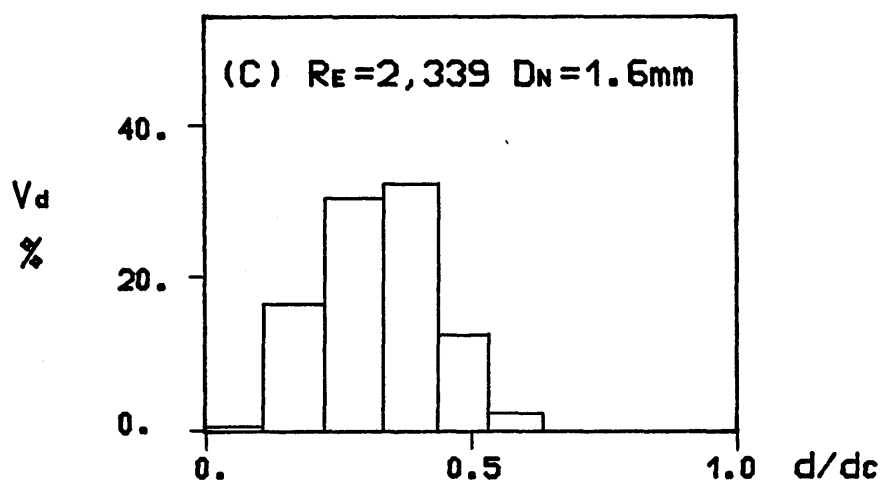
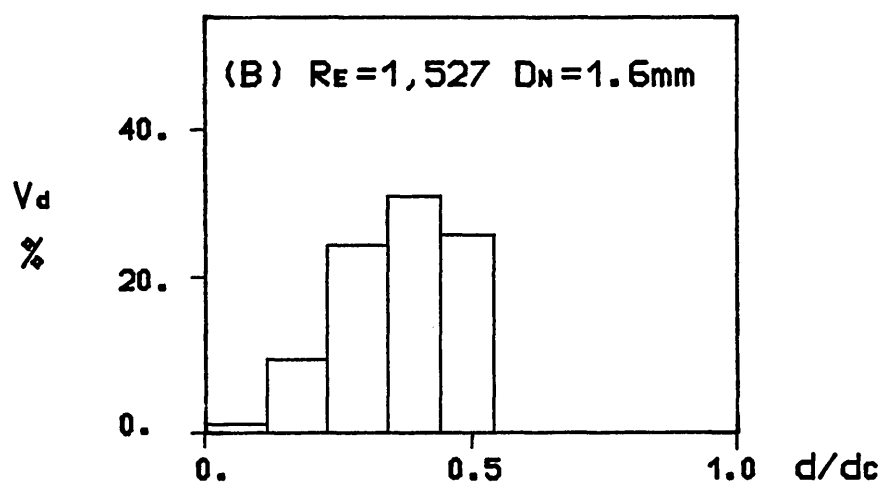
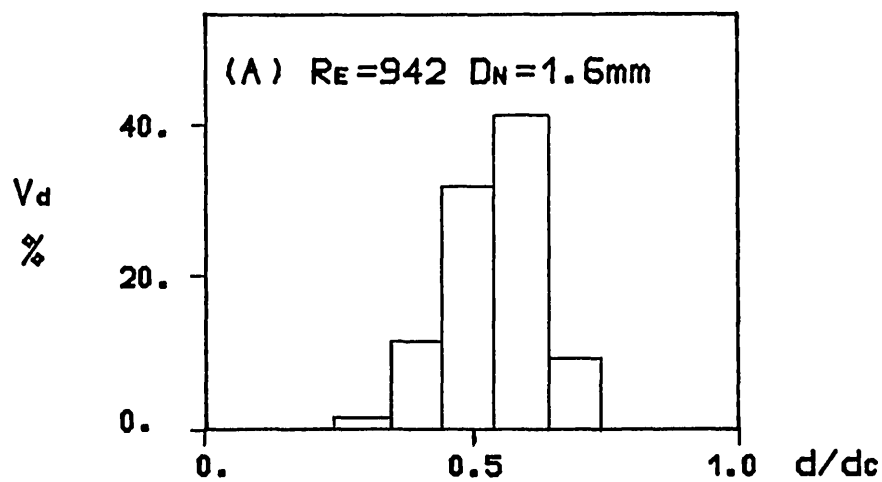


FIG.6.44-4(1) VOLUME PERCENTAGE OF DROPLETS
IN DIAMETER-SIZE GROUPS IN A
CROSS-SECTION: $Z=50D_N-60D_N$

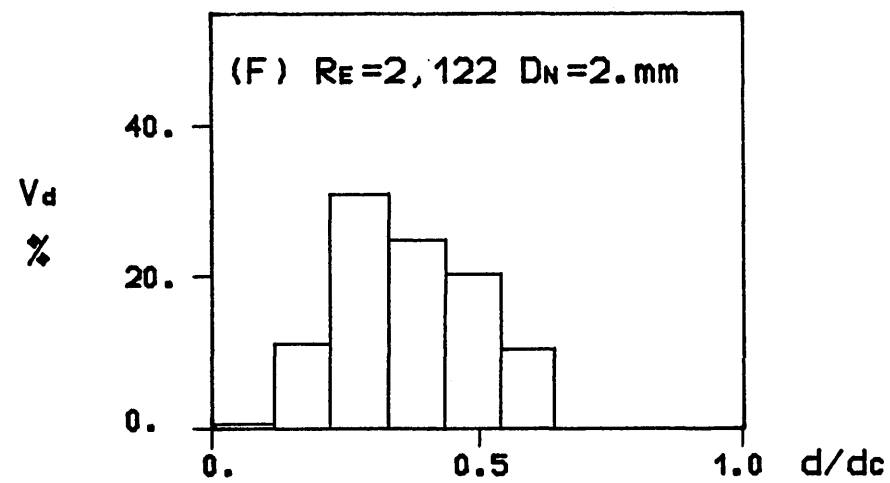
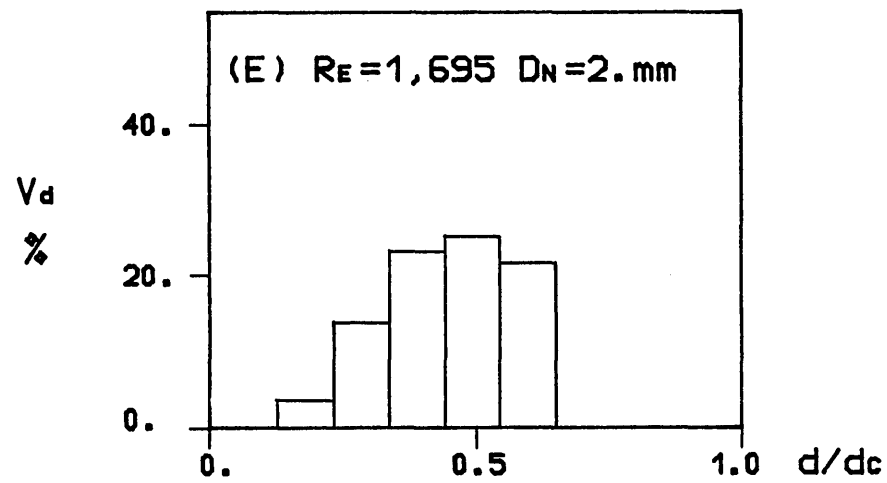
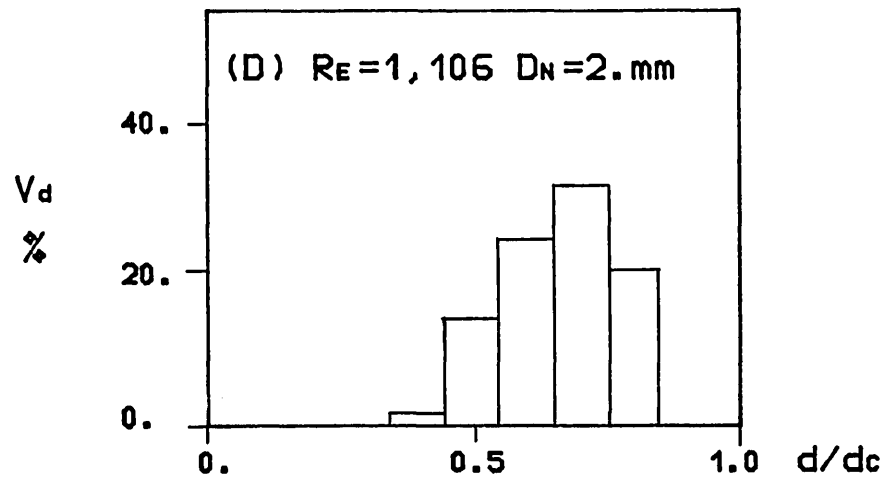


FIG.6.44-4(2) VOLUME PERCENTAGE OF DROPLETS
IN DIAMETER-SIZE GROUPS IN A
CROSS-SECTION: $Z=50D_N-60D_N$

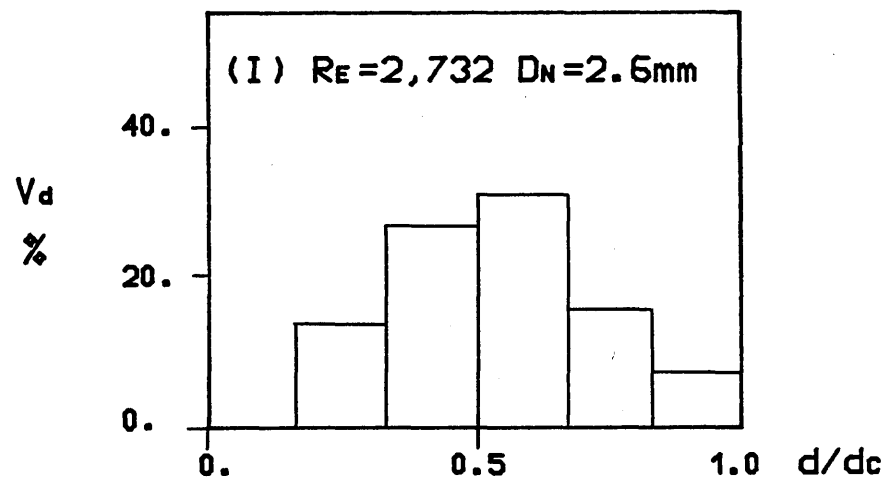
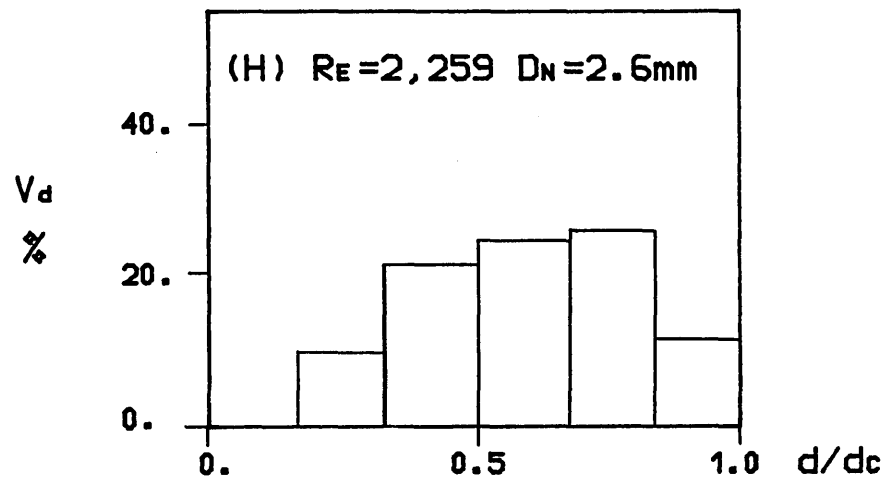
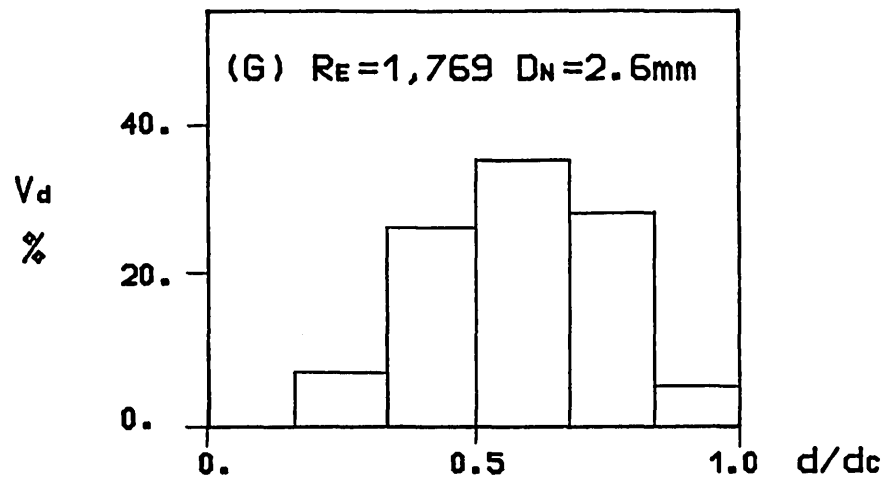


FIG.6.44-4(3) VOLUME PERCENTAGE OF DROPLETS
IN DIAMETER-SIZE GROUPS IN A
CROSS-SECTION: $Z = 50D_N - 60D_N$

$$d_c = 2 \left(\frac{\sigma}{g\Delta\rho} \right)^{0.5} \quad (6.44-1)$$

where σ is the interfacial tension;

g is the gravitational acceleration;

$\Delta\rho$ is the density difference between the two fluids.

For the kerosene-water flow d_c is 9.44 mm.

(3) Axial distribution of mean droplet diameter in cross-sections

The axial distribution of the mean droplet diameter in cross-sections was obtained as follows:

- Magnify a photographic negative on the screen with a transparent scaled sheet;
- Divide the whole jet region into a number of cross-sections;
- Divide the diameter of all droplets into a number of diameter-size groups;
- In each section, count the number of droplets falling into each group;
- Measure 24-30 negatives for each case;
- Calculate the Sauter mean droplet diameter for each section from: [Wallis, 1969]

$$\bar{d} = \frac{\int_0^{d_{\max}} d^3 p(d) dd}{\int_0^{d_{\max}} d^2 p(d) dd} \quad (6.44-2)$$

where $p(d)$ is the probability of the droplet having diameter between d and $d+\delta d$, $\bar{d}$ is the mean diameter and $d_{\max}$ is the maximum diameter.

Fig. 6.44-5 presents the axial distributions of the mean droplet diameter in cross-sections for all the cases considered. In the figure, $\bar{d}_z$ is the mean diameter in cross-sections. Appendix C gives the detailed experimental data.

(4) Mean droplet diameter within whole jet region

By applying equation (6.44-2) to the whole jet region (about $100 D_n$ long and covering the whole jet width), the mean droplet diameter of a jet can be obtained. Fig. 6.44-6 presents the mean droplet diameters of all the cases under consideration, where $\bar{d}_j$ is the mean diameter of a jet.

6.5 Discussion

(1) Similarities between miscible and immiscible transient turbulent jets

SYMBOL	Re	D_N
+	942	1.6mm
x	1,527	1.6mm
◆	2,339	1.6mm
⊠	1,106	2. mm
⊙	1,695	2. mm
▶	2,122	2. mm
•	1,769	2.6mm
□	2,259	2.6mm
△	2,732	2.6mm

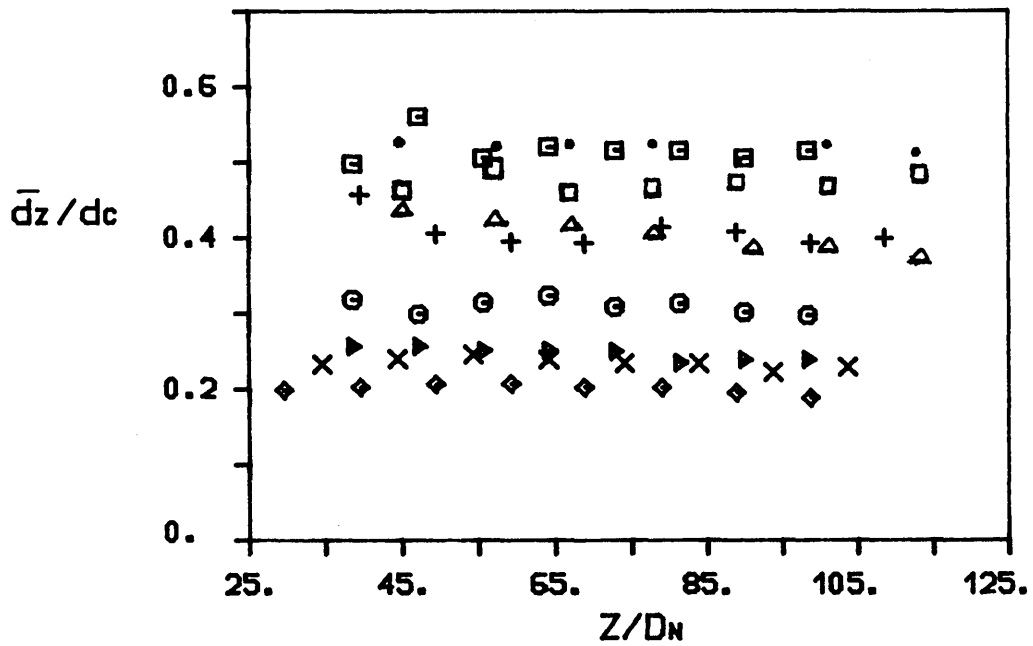


FIG.6.44-5 EXPERIMENTAL DATA OF THE AXIAL DISTRIBUTIONS OF THE MEAN DROPLET DIAMETERS OF KEROSENE-INTO-WATER STEADY-JETS

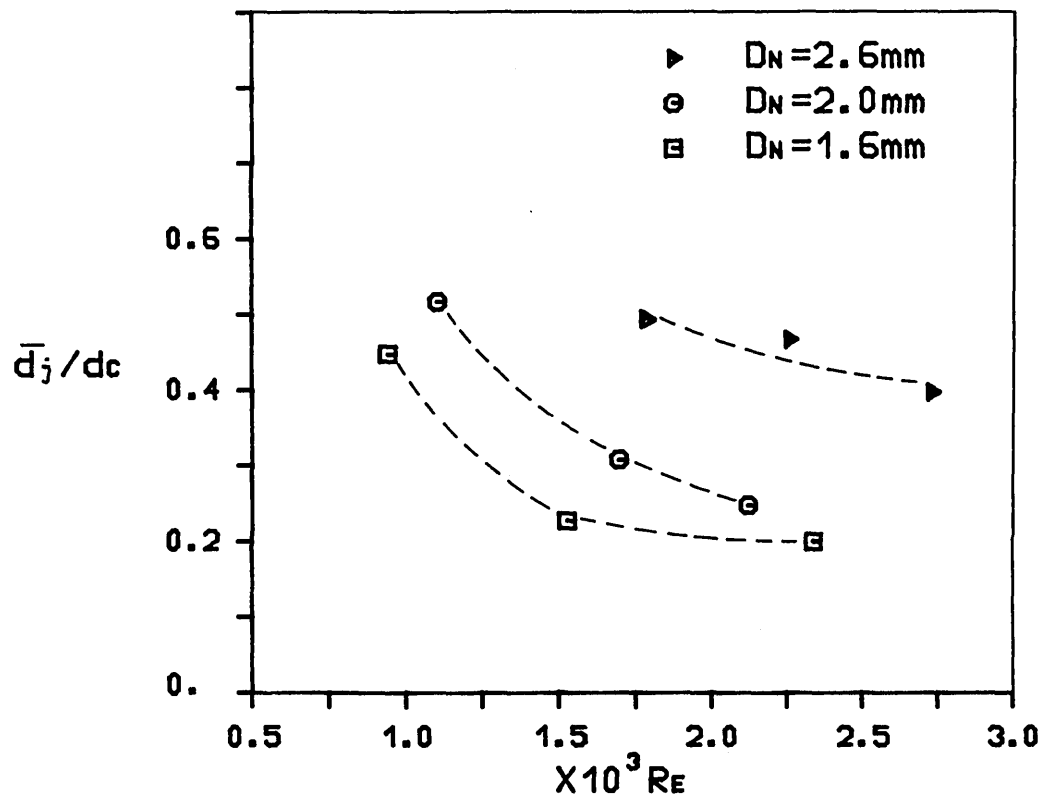


FIG. 6.44-6 EXPERIMENTAL DATA OF THE MEAN DROPLET DIAMETERS OF KEROSENE-INTO-WATER STEADY JETS

Fig. 6.42-1 shows the flow patterns of a miscible (brine-into-water) and an immiscible (kerosene-into-water) transient, turbulent jets. The following similarities between them can be found:

- (a) Both jets keep as stems in the immediate vicinity of the nozzle exit and then spread;
- (b) Both jets grow in the axial and radial directions while the surrounding fluid is entrained into the jet region;
- (c) The rates of spread of both jets are similar; and
- (d) Salient features are found at the bounding surfaces of the both jets. These features are that indentations of the surface grow and decay cyclically, and that each cycle leads to a substantial entrainment of surrounding fluid into the jet flow region.

The observed similarities between the miscible and immiscible jets suggest the possibility that both miscible and immiscible turbulent flows may be treated using same modelling assumptions.

(2) Tip-penetration rate measurement of transient jets

In the present study, tip-penetration rates of the brine-into-water and kerosene-into-water transient jets were measured. They were determined from a visual examination of the magnified photographic negatives. It can be found from Fig. 6.42-1, that the jet images have very clear edges. Therefore, the measurement of the height of the jet-tip from each negative does not involve large inaccuracies.

For the tip-penetration rate measurement, however, the errors were brought about due to the shortcomings of the apparatus. There was no synchronous device in the apparatus to open the throw-over valve and to switch on the camera simultaneously. In addition, opening the throw-over valve by hand further decreased the reproducibility. The errors are found less than 20% at the fully developed stages ($H_t/D_n > 50$).

(3) Volume flow rate measurement

The volume flow rate of the injected brine or kerosene was obtained by recording the variation with time of the ball position of the flowmeter. A video cassette recorder was used for this purpose. Since the variation of the ball position was recorded at a rather high speed (25 frames per second), and was then read from the tape frame by frame, good reproducibilities can be found in Figs. 6.43-1 to 6.43-6. It has also been noted that the Spline

Function used in the present study to fit the data is different from the polynomial function used in Hengrussamee's study (1975). The polynomial function is known to be less stable than the Spline Function.

(4) Droplet-size measurement

Direct photography used in the present study to measure the droplet size was adopted by many other authors in the past, e.g., Hayworth and Treybal (1950), Collins and Knudsen (1970). Kinter et al. (1961) presented a review of many techniques of photography. The long, tedious effort needed to obtain droplet-size distribution from photographs received some attention. For example, Adler et al. (1954) used the sweep of a narrow light beam and a photocell to measure droplet-size distributions. In addition, the following difficulties may arise when concentration of the droplets increases, or droplet size decreases: (a) the light transmittance decreases; (b) the higher magnification of the camera requires shorter exposure time; and (c) drop images may be distorted by drops between lens and plane of focus. In most cases of the present study, however, large droplets at low concentration presented no inherent photographic difficulties. Therefore, this method appears to be good for the present study.

Figs 6.44-4 (1), (2), and (3) show the volume percentages of the droplets in diameter-size groups in a cross-section for the cases considered. It can be found from these figures, that an increase in the *exit velocity* or a decrease in the nozzle diameter increases the volume percentage of the droplets having smaller size.

In Fig. 6.44-5, no considerable change in mean droplet diameter along the axis of each jet can be found. This indicates that almost all fragmentations take place within a short distance from the nozzle exit. The surface tension prevents the kerosene droplets from further fragmentation, while the interfluid friction between the droplet and surrounding water decreases the slip which is the dominant cause of further fragmentation.

Fig. 6.44-6 shows that the mean droplet diameter within a jet decreases with an increase in the *exit velocity* or a decrease in the nozzle diameter. This is expected, but needs to be further studied.

6.6 Closure

The experimental investigation of the present study can be summarized as follows:

- Similarities between miscible (brine-into-water) and

immiscible (kerosene-into-water) transient, turbulent jets have been found: both jets grow in a very similar manner;

- The tip-penetration rates of the two transient jets have been measured from the photographic negatives;
- The variation with time of the volume flow rate at the nozzle exit of the injected brine or kerosene of the transient jets has been recorded and then fitted by the Spline Functions;
- The mean droplet diameter of the kerosene-into-water steady jet has been obtained under various flow conditions (nozzle diameter, nozzle exit velocity); and
- The mean droplet diameters at different cross-sections of each jet do not change considerably along axis, while those in whole jet decrease with an increase in the *exit velocity* or a decrease in nozzle diameter.

These experimental data will be supplied for comparisons, boundary conditions, or modelling assumptions of computations. These will be presented in following two chapters.

CHAPTER 7 - COMPUTATION OF A BRINE-INTO-WATER TRANSIENT JET

7.1 Introduction

The previous chapter presented the experimental work. This chapter concerns the computations corresponding to the experiment of the brine-into-water transient jet.

Different from those in the air-into-air transient jet in Chapter 5, the boundary conditions at the jet inlet of this brine-into-water transient jet are time-dependent. In addition, this jet is affected by buoyancy, which is due to the density difference between the brine and water.

In the computations, the boundary conditions at the jet inlet are given from the experimental data obtained in Chapter 6. The tip-penetration rates of the jet are obtained from the computational outputs, and are then compared with the experimental data.

Since the empirical constants of the two-fluid assumptions did not exert any considerable effects on the results of the mean flow in the previous computations, only the effect of the density ratio of the injected brine to surrounding water on the results is tested.

The remainder of this chapter consists of seven sections.

The flow conditions which are to be simulated are presented in Section 7.2. Section 7.3 contains the computational details. Section 7.4 describes the definition of the jet-tip. Section 7.5 presents the computational results and their comparisons with the experimental data. Section 7.6 shows the effect of the density ratio. The work of this chapter is discussed in section 7.7 and finally, Section 7.8 gives the conclusion.

7.2 The flow considered

The brine-into-water transient jet which was experimentally studied in Chapter 6 is considered. This flow can also be briefly described by the following:-

- brine is injected from the start;
- velocity of the brine at the nozzle exit is developing, and is gradually attaining a constant;
- the surrounding tap water is initially at rest;
- the formed jet is a single-phase flow;
- the flow is transient;
- buoyancy exists in the positive axial direction; and
- axial symmetry prevails.

Although the velocity of the injection is gradually increasing from zero, the flow is presumed to be a turbulent one. The constant velocity at the nozzle exit,

which the injection gradually reaches, is taken to be the nozzle exit velocity W_i in equation (2.41-1) to calculate the effective viscosity. These assumptions will be discussed later.

The conditions of the flows which are to be predicted are given in Table 7.2-1 below.

Case	D_n (mm)	V_i ($10^{-6} \text{ m}^3/\text{sec.}$)	W_i (m/sec.)	Re
7-1	1.6	4.21	2.09	3,344
7-2	1.6	8.42	4.18	6,688
7-3	2.0	4.21	1.34	2,680
7-4	2.6	8.42	1.59	4,108

Table 7.2-1 Flow conditions considered

In table 7.2-1, V_i , W_i and Re are their values when the flows reach a steady-state.

7.3 Computational details

7.31 Solution procedure and calculation domain

As the air-into-air transient jet, the brine-into-water transient jet is an elliptic and transient flow. The governing equations with modelling assumptions given in Chapter 2 are numerically solved by the PHOENICS-84 computer code.

Again, fluid-1 refers to the turbulent (rotational) fluid and fluid-2 refers to the non-turbulent (irrotational) fluid in the computations.

Because the flow conditions of the brine-into-water transient jet are similar to those of the air-into-air transient jet. The integration domain in this chapter is almost same as that in Chapter 5. Only difference between the two domains is the length of the domain. In Chapter 5, the length was 100 D_n while that in this chapter is 125 D_n or 150 D_n . It is because that the flow fields to be predicted in this chapter are longer than those in Chapter 5.

The sketch of the integration domain for the computations of this chapter is shown in Fig. 7.31-1.

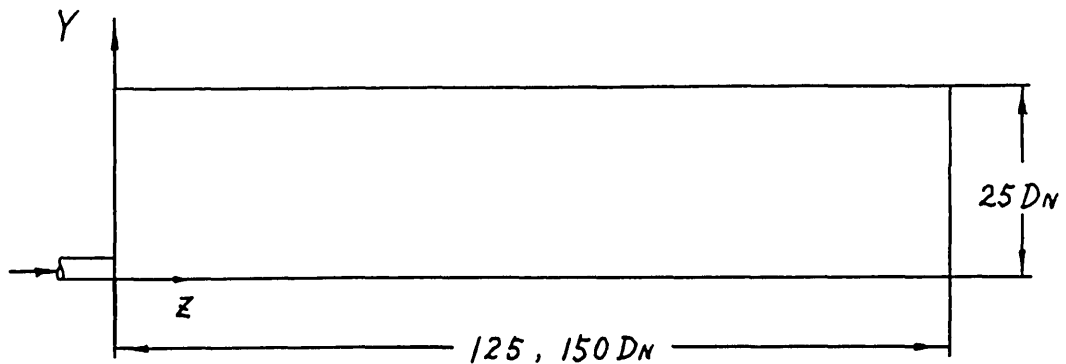


FIG. 7.31-1 THE CALCULATION DOMAIN FOR THE
BRINE-INTO-WATER TRANSIENT JET

7.32 The grid and time-step

To obtain an economic and accurate solution the effects of the number of the grid-node and time-step on the prediction have been investigated.

Following those in Chapter 5, non-uniformly distributed grid-nodes and time-steps have been tested in the investigation. Smaller intervals were set for the radial nodes near the axis, and for the axial nodes near the nozzle exit, while smaller time-steps were set for the initial period of the computations.

Case 7-1 of Table 7.2-1 was taken to be the testing flow,

the results of the investigation can be summarized as follows:

Grid

400, 625, 700, 800, and 875 grid-nodes, i.e. 20 x 20, 25 x 25, 25 x 28, 25 x 32, and 25 x 35 nodes in the radial and axial directions have been tested.

Smaller intervals were set for the radial grid-nodes near the axis. For the 25 radial nodes, from the axis outwards, the first 3 nodes occupied 2% of the domain, second 3 nodes occupied 3% of the domain, next 9 nodes occupied 18% of the domain, next 6 nodes occupied another 18% of the domain, and final 4 nodes occupied the rest of the domain.

The axial grid-nodes were distributed according to the power-law which was given in equation (5.32-1).

With these grid-nodes and expanding time-steps from 0.002 sec. to 0.02 sec., Fig. 7.32-1 shows the computed heights of the jet-tip for case 7-1 at $t=0.8$ sec.. (The definition of the location of the jet-tip will be given later). It has been confirmed that the 25 x 32 grid-nodes which were preferred in Chapter 5 are also considered as the best choice for this computation.

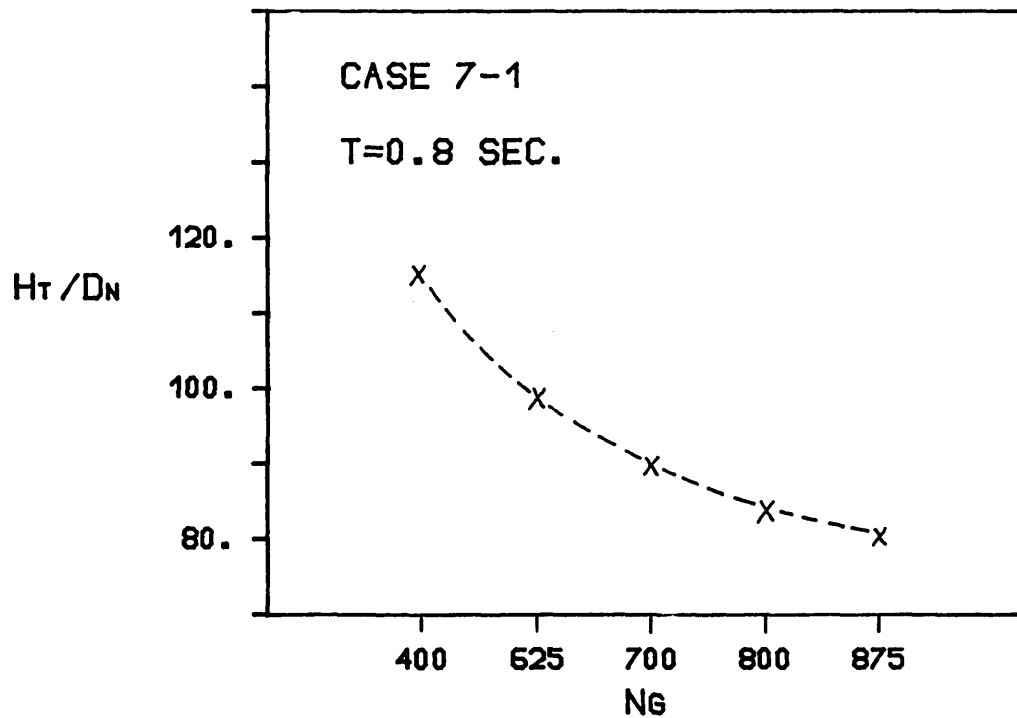


FIG. 7.32-1 EFFECT OF GRID-NODE NUMBER ON THE HEIGHT OF THE JET-TIP

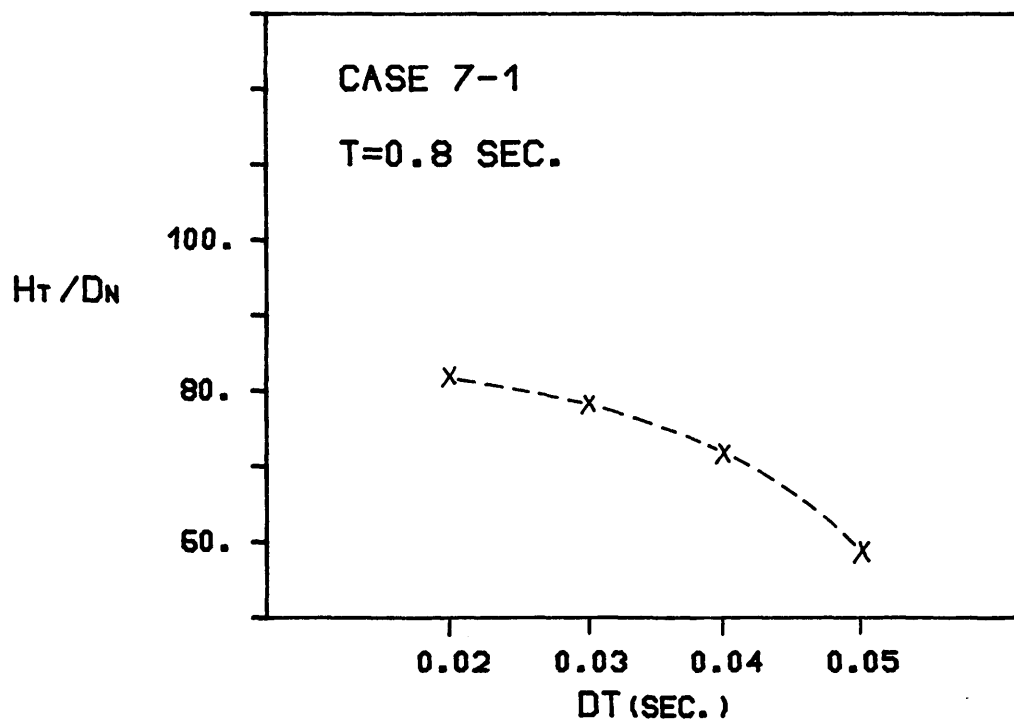


FIG. 7.32-2 EFFECT OF TIME-STEP ON THE HEIGHT OF THE JET-TIP

Time-step

Expanding time-steps have been tested in the investigation. The initial time-step was given a small value of 0.002 sec., it was then increased by a factor 1.1 at every forward time step, until the assigned values of 0.05, 0.04, 0.03, or 0.02 sec. was reached. Fig. 7.32-2 shows the computed heights of the jet-tip of case 7-1 at $t=0.8$ sec.. From the figure, the expanding time-steps from 0.002 sec. to the maximum of 0.02 sec. are considered as the best choice.

The non-uniformly distributed 25x32 grid-nodes and expanding time-steps from 0.002 sec. to the maximum of 0.02 sec. were used to produce the main results of this chapter.

7.33 Boundary conditions

Fig. 7.33-1 shows the boundaries of the integration domain for the problem of this chapter.

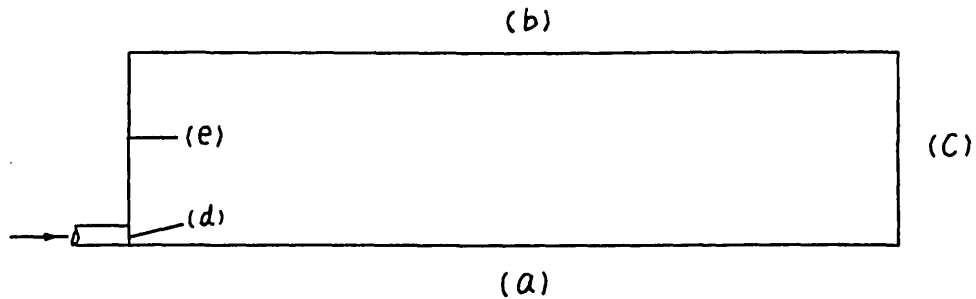


FIG. 7.33-1 THE BOUNDARIES OF THE CALCULATION
DOMAIN FOR THE BRINE-INTO-WATER
TRANSIENT JET

These boundaries are specified as follows:

- Boundary (a) is the inner boundary. This is a symmetrical axis so that a zero-flux boundary condition is specified;
- Boundaries (b) and (c) are the outer and downstream boundaries of the domain. In the experiment, the brine was injected into the tap water in a tank. Since the dimensions of the tank are much larger than those of the domain, the effect of the tank walls can be neglected. Consequently, fixed-pressure boundary conditions are specified.

At boundary (d), i.e. the jet inlet,

- only turbulent fluid is present, i.e. intermittency factor is equal to 1;
- top-hat profiles are specified for the streamwise velocity of the turbulent fluid and the intermittency factor;
- no radial velocity components exist;
- the streamwise velocity of the turbulent fluid is equal to that of the injected brine; and
- the volume flow rate as well as the velocity of the injected brine are developing, and are given by the "Spline Functions" obtained in Chapter 6.

At boundary (e),

- as for boundaries (b) and (c), a fixed-pressure boundary condition is specified to simulate the experimental condition.

7.34 Initial conditions

In the experiment, the brine was injected from the start, and the water in the tank was initially stagnant. Zero pressure, zero velocity components of both fluids, volume fraction $r_1=0$, $r_2=1$ are specified over the whole domain as the initial conditions of the computations to simulate the

experimental conditions.

7.35 Convergence

It was necessary to use the relaxation facilities built in the code to obtain a converged solution. The "FALSDT" was used for the velocity components, while the "LINRLX" was for the volume fraction of two fluids. The detailed descriptions of these relaxation facilities can be found in "PHOENICS-84 reference handbook" (Rosten and Spalding, 1984).

15 sweeps were required at each time step. The order of the whole-field sum of the absolute value of the volume flow residuals at the end of each time step is 10^{-7} .

Table 7.35-1 summarizes all the information about the convergence of the computation.

Variable	Value
Grid	25 grid-nodes in Y-direction 32 grid-nodes in Z-direction
Type of grid spacing	non-uniform; finer near the axis and inlet
Time-step	expanding; from 0.002 sec. to 0.02 sec. by factor 1.1
Number of iterative sweeps at each time-step	15
Relaxation	FALSDT: 0.1 for V_1 , V_2 , W_1 , W_2
	LINRLX: 0.1 for r_1 , r_2
Whole-field sum of ABS(vol. flow residuals)	1.251×10^{-7}

Table 7.35-1 Computational details for the
brine-into-water transient jet

7.36 Computer storage and computer time

As those for the air-into-air transient jet, the computer

storage and computer time needed for this flow are much more than those for a parabolic and steady flow.

Table 7.36-1 displays the essential incore space and the computer time for different grid-node numbers used for the whole domain.

Grid NYXNZ	Computer storage	Computer time per time-step (sec.)
20x20	10,396	118.97
25x25	15,492	175.0
25x28	17,124	200.0
25x32	19,278	224.66
25x35	20,886	246.02

Table 7.36-1 Computational requirement for the
brine-into-water transient jet
(Computer used: Perkin Elmer
3220)

7.4 Definition of jet-tip

The jet-tip is defined as the axial location of the horizontal plane where the mass flow rate of the turbulent fluid is equal to 1% of that at the nozzle exit.

The mass flow rate of the turbulent fluid across each horizontal plane is calculated at each time step. This mass flow rate M can be expressed as:

$$M = 2\pi \int_0^R w_1 \rho_1 r_1 y dy \quad (7.4-1)$$

where R is the radius of the integration domain.

7.5 Computational results of tip-penetration rate

In this chapter, the main results presented and compared with the experimental data are the penetration rates of the jet-tip, i.e. the increases with time in the height of the jet-tip. They are obtained from the field values of the computational outputs via equation (7.4-1).

The flow conditions of the four cases which have been computed were summarized in Table 7.2-1, while the experimental data were obtained in Chapter 6. The computational results of the tip-penetration rate are now deduced and presented.

According to the definition in Section 7.4, the location of the jet-tip at each time step can be deduced as

follows:

- The mass flow rate of the turbulent fluid across each horizontal plane is calculated from equation (7.4-1);
- Distribution of the mass flow rate along the centre-line is then obtained; and
- Jet-tip is given by finding the location of the horizontal plane where the mass flow rate is 1% of that at the nozzle exit.

Figs. 7.5-1 to 7.5-4 show the computational and experimental tip-penetration rates for the four cases. In these figures, dimensionless variables H_t/D_n and T_{xWi}/D_n are used again, the solid lines are the computational results, while the symbols are the data.

7.6 Effect of density ratio of injected brine to surrounding water

To show the effect of the density ratio of the injected brine to the surrounding water Case 7-1 was again taken as the flow to calculate. The effect on the predicted penetration rate of the jet-tip and the predicted profiles of the streamwise velocities and intermittency factor at $Z/D_n = 36$ and $T = 0.8$ sec. are presented in Figs. 7.6-1 and 7.6-2, 3 respectively.

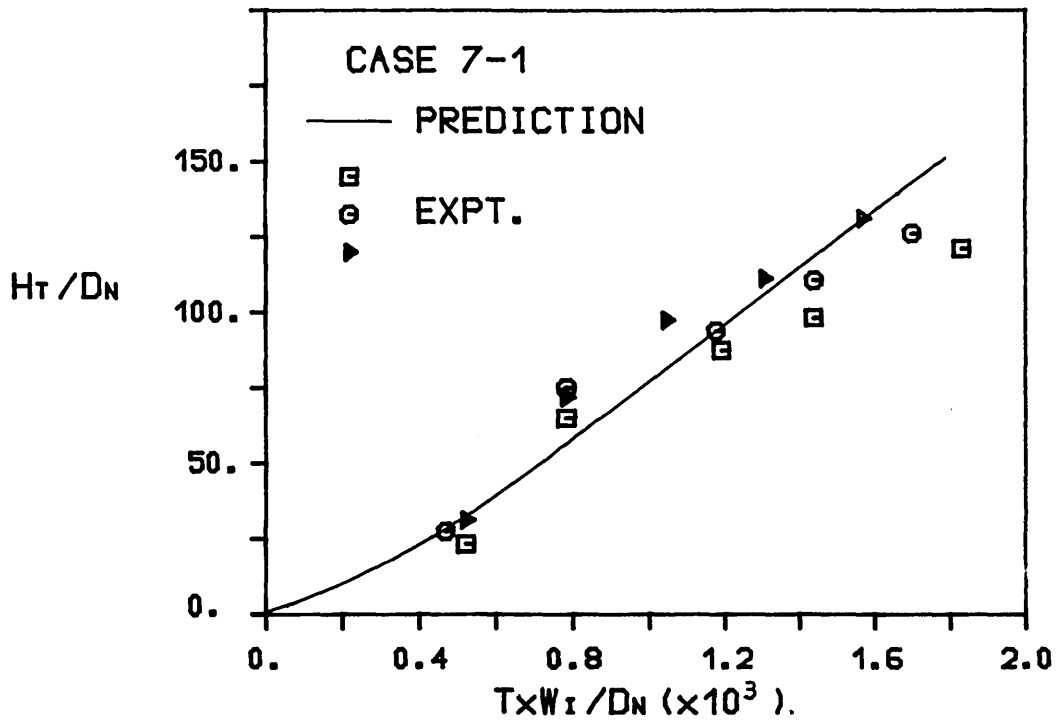


FIG. 7.5-1 TIP-PENETRATION RATE OF A BRINE-INTO-WATER TRANSIENT JET: CASE 7-1

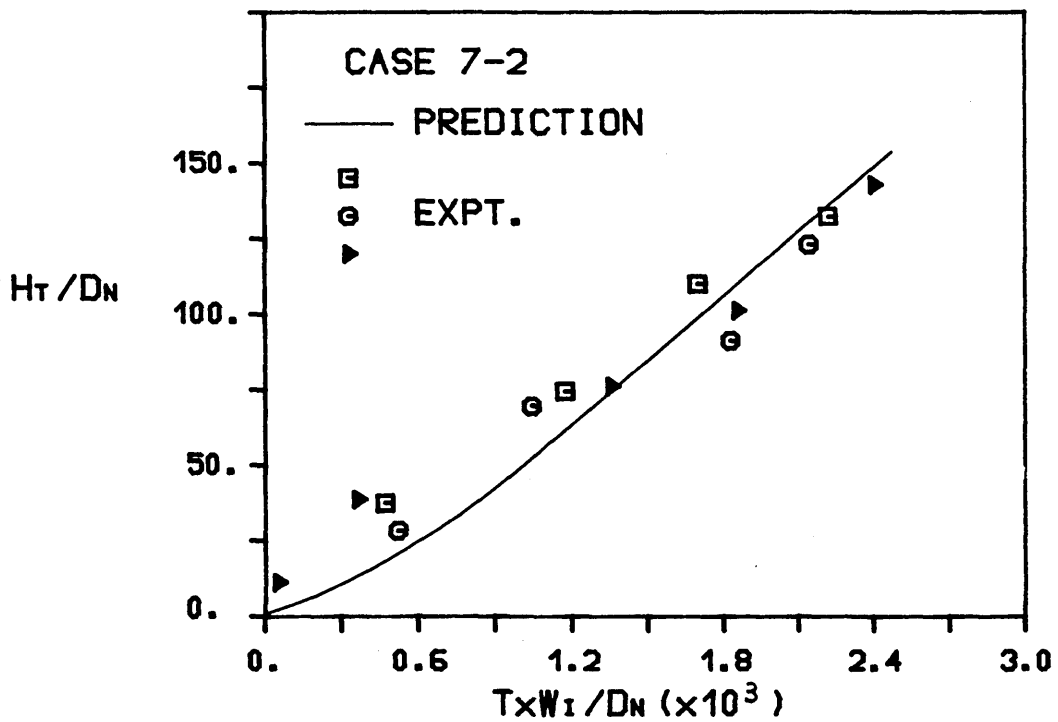


FIG. 7.5-2 TIP-PENETRATION RATE OF A BRINE-INTO-WATER TRANSIENT JET: CASE 7-2

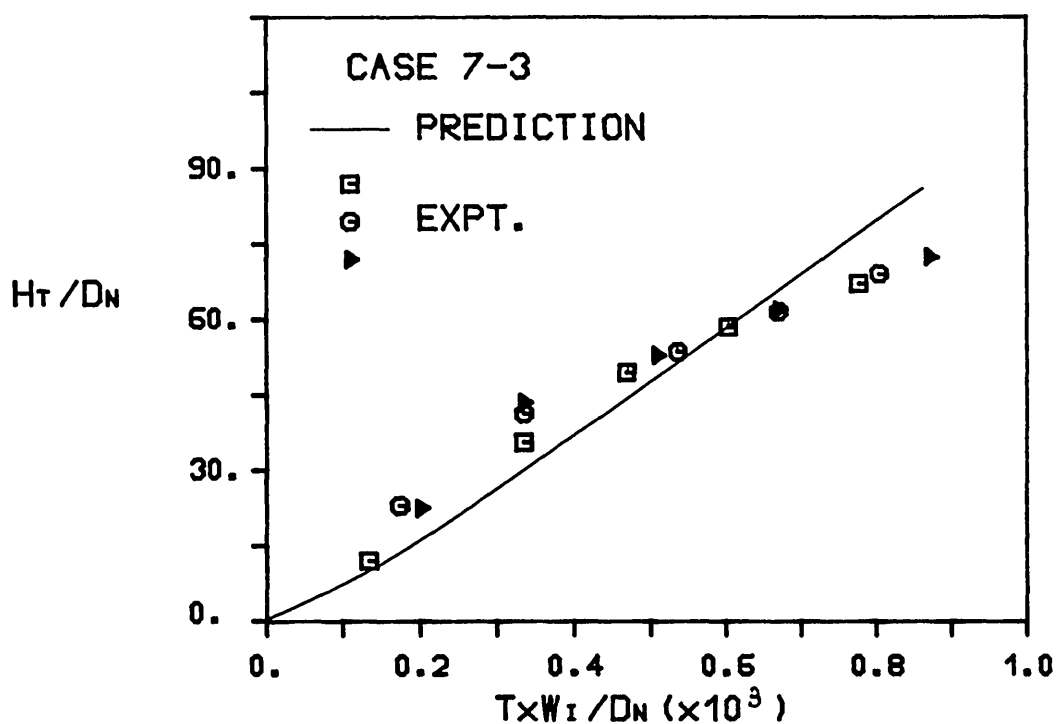


FIG. 7.5-3 TIP-PENETRATION RATE OF A BRINE-INTO-WATER TRANSIENT JET: CASE 7-3

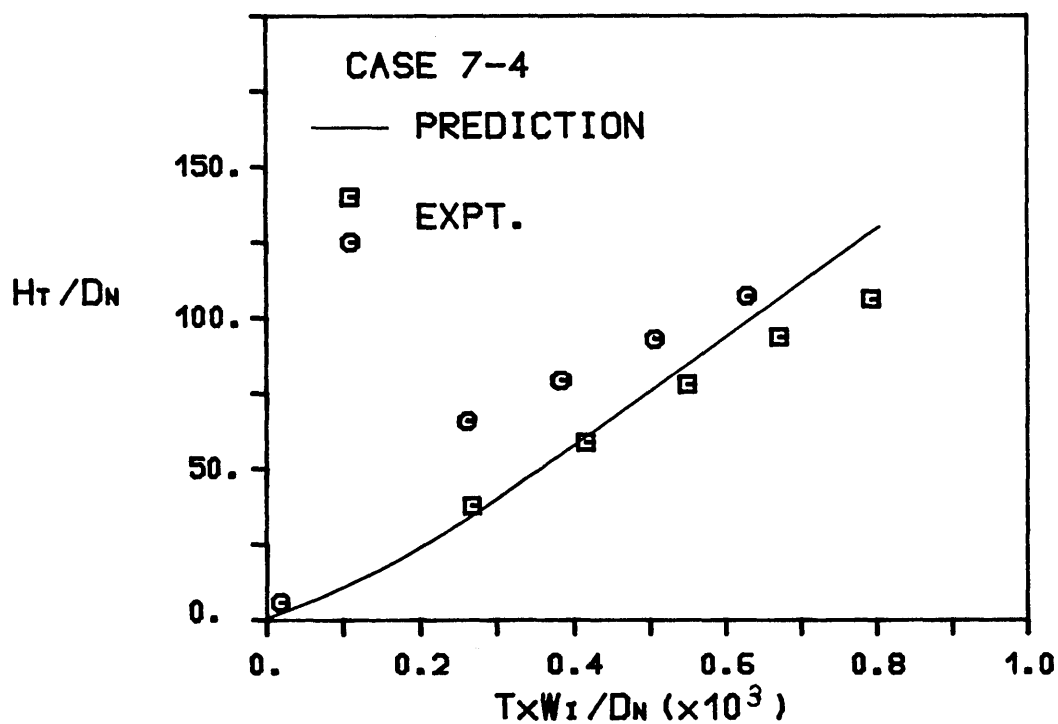


FIG. 7.5-4 TIP-PENETRATION RATE OF A BRINE-INTO-WATER TRANSIENT JET: CASE 7-4

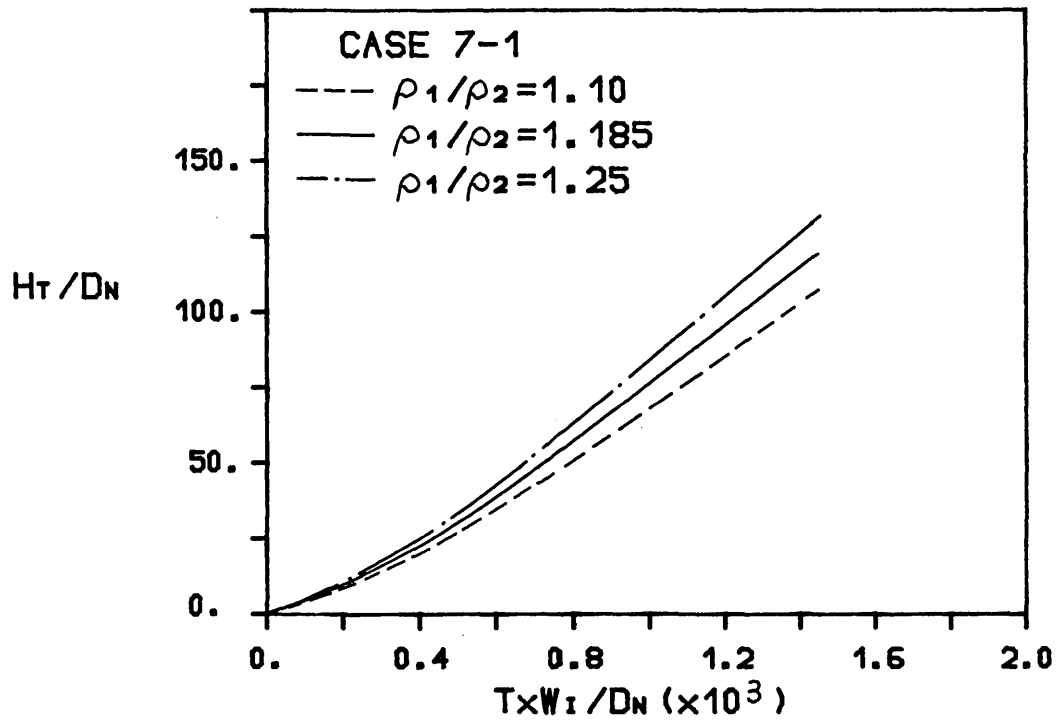


FIG. 7.6-1 EFFECT OF ρ_1/ρ_2 ON THE PREDICTED TIP-PENETRATION RATE OF A BRINE-INTO-WATER TRANSIENT JET

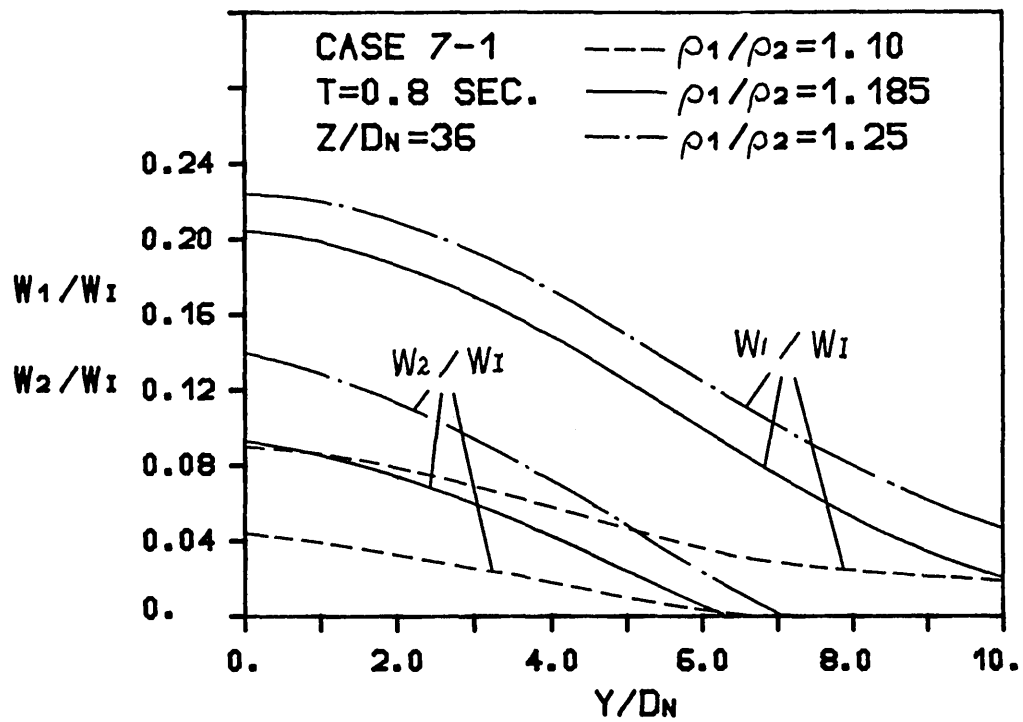


FIG. 7.6-2 EFFECT OF ρ_1/ρ_2 ON THE PREDICTED PROFILES OF STREAMWISE VELOCITIES

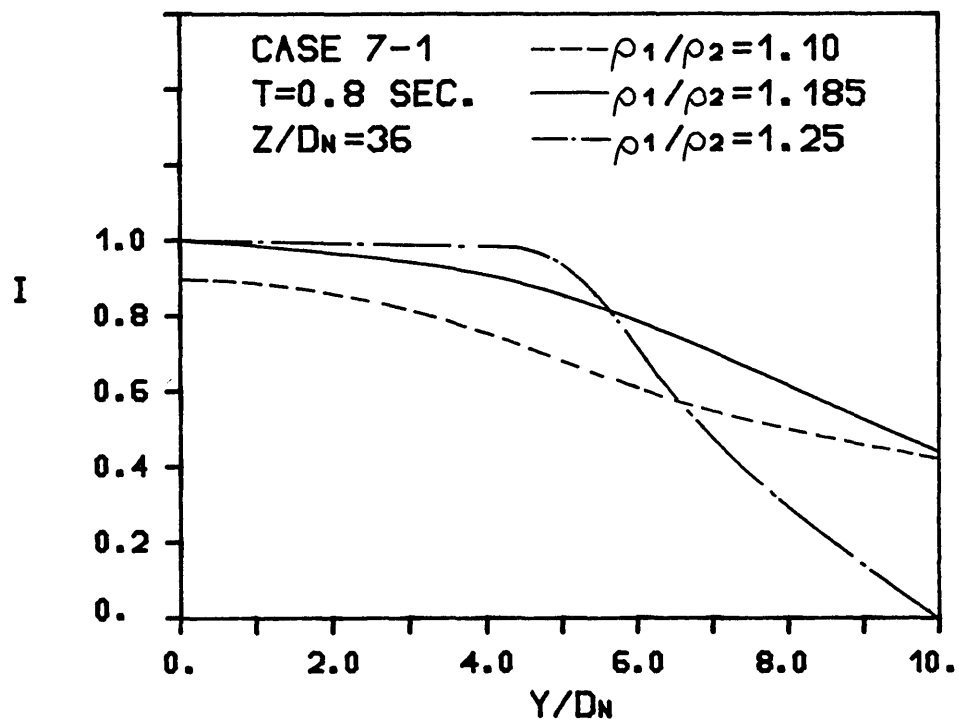


FIG. 7.6-3 EFFECT OF ρ_1/ρ_2 ON THE PREDICTED PROFILE OF INTERMITTENCY FACTOR

7.7 Discussion

In this chapter, the tip-penetration rate of the brine-into-water transient jet has been predicted, and the prediction has been compared with the experimental data.

In the experiment of the present study, the locations of the jet-tip were determined from the visual examination of the magnified photographic negatives. The predicted ones were obtained from the computational outputs according to the prescribed definition given in Section 7.4. It is worth noting, that these methods are different from what other authors have used. Abramovich and Solan (1973) and Witze (1980) defined the jet-tip as the location where the transient streamwise velocity reached 70% of its steady state value in their experiments. In the calculations of Kuo (1982), the same definition was used to determine the predicted location of the jet-tip. It is known that the definition of the jet-tip is a function of the experimental techniques used to measure it. In the present study, the definition by the axial distribution of the mass flow rate of the turbulent fluid is adopted, and is considered corresponding with the experimental observation.

For convenience of discussion, the development of the transient jet is divided into three periods, Period (a), (b), and (c). Each of them equally shares the time of

whole process investigated.

From the comparisons between the predictions and experiments for the tip-penetration rate in Figs. 7.5-1 to 7.5-4, it can be found that agreements are generally satisfactory. But the computations underpredicted it in Period (a) and overpredicted it in Period (c).

As stated in Section 7.2, the transient jet was assumed to be turbulent flow, and the constant velocity at the nozzle exit, which the injection gradually reached, was taken as the W_i in equation (2.41-1) to give the turbulent viscosity. As a matter of fact, the nozzle exit velocity was increasing from zero, while the flow was developing from laminar to turbulent. Therefore, the turbulent viscosity was overestimated, and this brought more viscous transport of the momentum in the radial direction than what actually happened. This was particularly severe in Period (a) during which the nozzle exit velocity was much lower than the constant one.

In the model, the mixing length expressed in equation (2.42-4) was taken to be the mean fragment size. This mixing length depends upon only the axial distance, the longer the distance the larger the size. However, the jet-tip is just the leading front of a flow, it can't form a larger and larger fragment. This implies that the model

overestimated the increase in the fragment size of the jet-tip at far field from the exit. Therefore, the interfluid mass transfer from the non-turbulent fluid to the turbulent one, and the interfluid friction factor were underestimated. These can be found from equations (2.42-1) and (2.42-7). The penetration rate of the jet-tip at the far field may thus be overestimated.

As expected, an increase in the density ratio of the injected brine to the surrounding water leads to an increase in the predicted penetration rate of the jet-tip. This is shown in Fig. 7.6-1.

7.8 Conclusion

The tip-penetration rate of the brine-into-water transient jet with time-dependent boundary conditions has been predicted in this chapter.

The performance of the mathematical model for this flow can be summarized as follows:

- (1) The same modelling assumptions and empirical constants as those for the air-into-air steady or transient jets are acceptable for this flow.
- (2) In general, agreement between the prediction and data of the tip-penetration rate is satisfactory.

- (3) The computations underpredict the penetration rate in the initial period and overpredict it in the final period.

In addition to the refinements of the modelling assumptions suggested in Chapters 4 and 5, more efforts of using other techniques in the experiment, and testing various definitions of the jet-tip in the computation are required to further study the phenomena.

CHAPTER 8 - COMPUTATION OF A KEROSENE-INTO-WATER TRANSIENT JET

8.1 Introduction

In Chapters 4, 5 and 7, the round jets of miscible fluids with various flow conditions were numerically simulated using the mathematical model described in Chapter 2. For miscible flows, the model artificially treats them as random interspersions of two fluids by making use of mathematical apparatus developed primarily for a two-phase flow. The computations of this chapter are concerned with the simulation of a genuine two-phase transient jet.

In Chapter 6, the experiments for the kerosene-into-water transient and steady jets were presented. In these experiments, the penetration rate of the jet-tip, development of the volume flow rate of the kerosene at the nozzle exit, and the droplet size of the jet under steady-state conditions were measured.

The kerosene-into-water jet is different from the air-into-air and brine-into-water jets. The two fluids of the kerosene-into-water jet are immiscible forming a genuine two-phase flow.

Computations corresponding to the experiment of this kerosene-into-water transient jet are now described. The

same constant-effective-viscosity model is used to calculate the turbulent viscosity, while modifications of the two-fluid assumptions described in Chapter 2 are made for the present computations. As in Chapter 7, the quantity of interest is the penetration rate of the jet-tip. The computational results of the penetration rate are obtained via a prescribed definition, and are then compared with the experimental data obtained in Chapter 6.

Computations have also been carried out to show the effects of an empirical constant C_f of the two-fluid model, and the density ratio of the two fluids.

The remainder of this chapter consists of eight sections. Section 8.2 gives the flow conditions to be computed, while modifications of the two-fluid assumptions are described in Section 8.3. The computational details are contained in Section 8.4. Section 8.5 defines the jet-tip. Section 8.6 presents the computational results and comparisons with the experimental data of the penetration rate. Section 8.7 demonstrates the effects of an empirical constant C_f and the density ratio and finally, Section 8.8 is devoted to the discussion followed by the conclusion in Section 8.9.

The listings of the "Q1" and "GROUND" files of the PHOENICS computer code for the computations of this

chapter are given in Appendix D.

8.2 The flow considered

The kerosene-into-water transient jet with the experimental conditions given in Chapter 6 is considered. This flow can also be described by the following:

- kerosene is injected from the start;
- velocity of the kerosene at the nozzle exit is developing, and is gradually attaining a constant;
- the surrounding tap water is initially at rest;
- the formed jet is a two-phase flow;
- the flow is transient;
- buoyancy exists in the positive axial direction; and
- axial symmetry prevails.

As in Chapter 7, although the flow is developing from laminar to turbulent, it is assumed to be a turbulent flow. The constant velocity at the nozzle exit, which the injection gradually reaches, is taken as the nozzle exit velocity W_i in equation (2.41-1) to calculate the effective viscosity.

Table 8.2-1 summarizes the flow conditions which are to be computed.

Case	Dn (mm)	Vi (10^{-6} m ³ /sec.)	Wi (m/sec.)	Re
8-1	1.6	3.77	1.88	1,527
8-2	1.6	5.79	2.88	2,339
8-3	2.0	6.57	2.09	2,122
8-4	2.6	11.01	2.07	2,732

Table 8.2-1 Flow conditions considered

8.3 Modifications of two-fluid assumptions

To simulate a genuine two-phase flow of this kerosene-into-water transient jet, the two-fluid assumptions presented in Chapter 2 have been modified. These modifications can be briefly described as follows:

- The injected kerosene and surrounding water are defined as fluid-1 and fluid-2 respectively;
- No interfluid mass transfer exists between the two fluids;
- In the expression of the interfluid friction factor, equation (2.42-7), $C_f=0.1$ is adopted, while a high

value, $C_f=500$ is set at the first axial station;

- Mean droplet diameter $\bar{d}_j$ of the kerosene-into-water steady jet is taken as the mean fragment size L in the modelling assumptions. This mean droplet diameter was experimentally obtained in Chapter 6.

These will be discussed later in Section 8.8.

8.4 Computational details

8.41 Solution procedure and calculation domain

As the air-into-air transient jet in Chapter 5 and the brine-into-water transient jet in Chapter 7, this kerosene-into-water transient jet is an elliptic and transient flow. The PHOENICS-84 computer code is used for solving the governing equations and modelling assumptions given in Chapter 2 and Section 8.3.

The integration domain of this chapter is almost same as that in Chapter 7. The length of the domain is 125 Dn which is considered to be remote sufficiently from the predicted flow field for all cases computed in this chapter.

8.42 The grid and time-step

Following that in Chapter 7, non-uniformly distributed

grid-nodes were adopted in the computations; smaller intervals were set for the radial grid-nodes near the axis, and for the axial grid-nodes near the nozzle exit. Equal time-steps were used in this chapter.

Case 8-1 of Table 8.2-1 was taken as the testing flow, and the results of the investigation for the grid-node and time-step number effect are now presented.

Grid

The grid-nodes tested in this chapter are exactly same as those in Chapter 7. With these grid-nodes and a constant time-step of 0.02 sec., Fig. 8.42-1 shows the computed heights of the jet-tip for Case 8-1 at $t=0.6$ sec.. (The definition of the location of the jet-tip will be given later.) From the comparison, it has been again confirmed that the 25x32 grid-nodes are considered as the best choice for the present computations. The intervals of these nodes can be seen in Section 7.32.

Time-step

Equal time-steps of 0.05, 0.04, 0.03, and 0.02 sec. have been tested in the investigation. Fig. 8.42-2 shows the computed heights of the jet-tip for Case 8-1 at $t=0.6$ sec.. From the figure, it can be found that the time-step

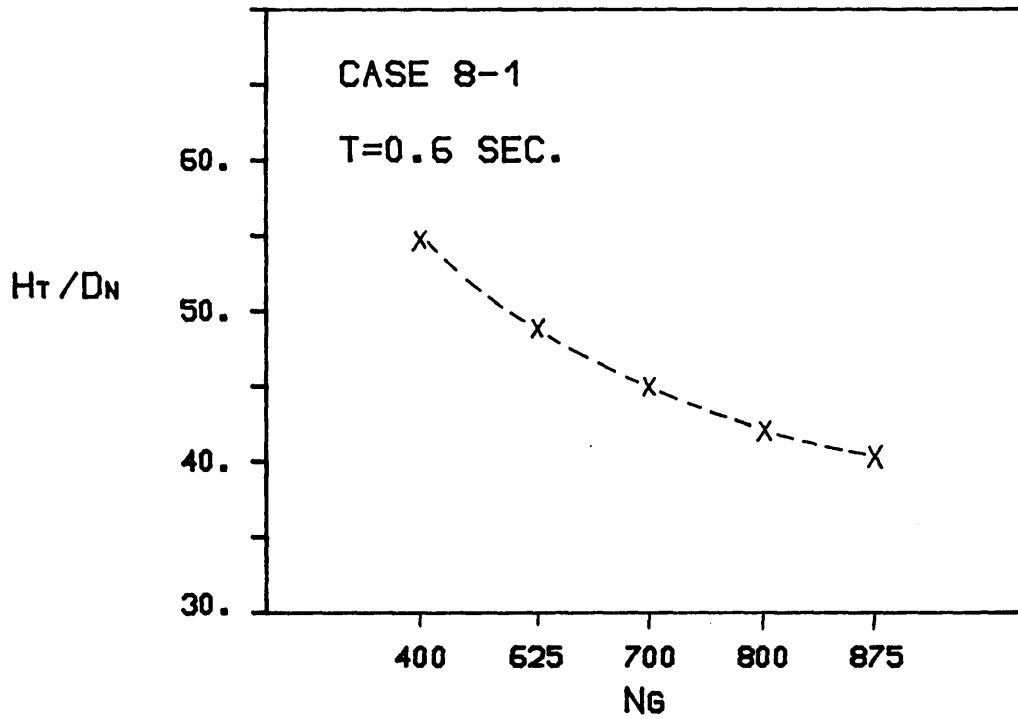


FIG. 8.42-1 EFFECT OF GRID-NODE NUMBER ON THE HEIGHT OF THE JET-TIP

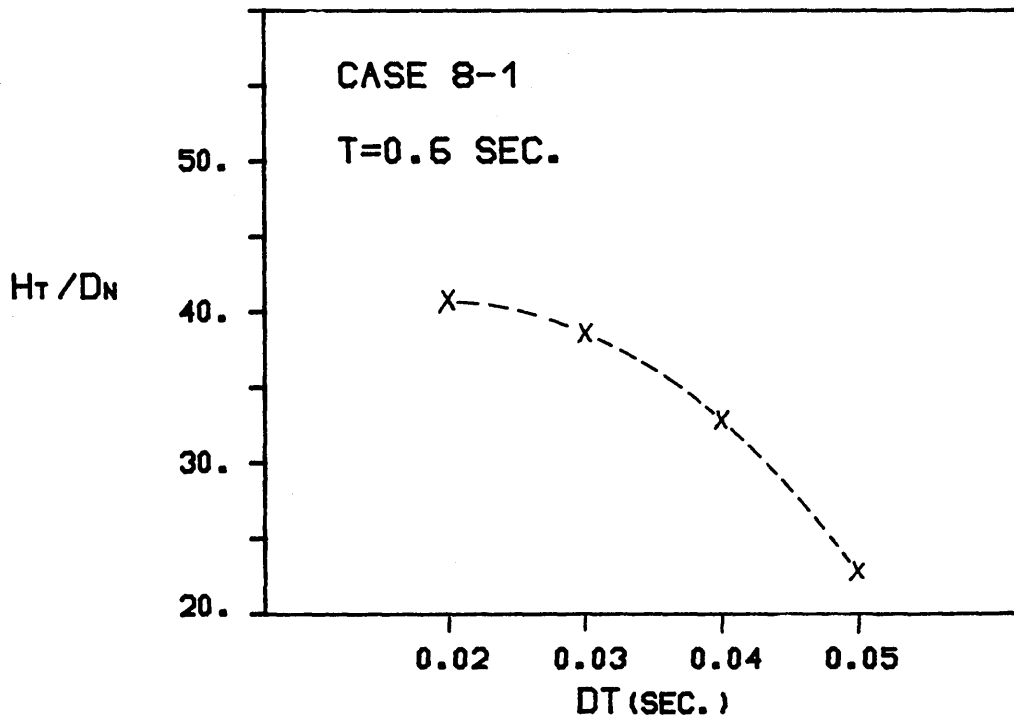


FIG. 8.42-2 EFFECT OF TIME-STEP ON THE HEIGHT OF THE JET-TIP

of 0.02 sec. is preferred.

Non-uniformly distributed 25x32 grid-nodes and equal time-step of 0.02 sec. have been used to produce the main results of this chapter.

8.43 Boundary conditions

The boundary conditions of this chapter are specified in a same way as that in Chapter 7, except those at the jet inlet.

At the jet inlet,

- only fluid-1 (kerosene) is present, i.e. volume fraction of fluid-1 is equal to 1;
- top-hat profiles are specified for the streamwise velocity and volume fraction of fluid-1;
- no radial velocity components exist;
- the volume flow rate as well as the velocity of the injected fluid-1 are developing with time, and are given by the "Spline Functions" obtained in Chapter 6.

8.44 Initial conditions

Similar to those in Chapter 7, zero pressure, zero velocity components of two fluids, volume fractions $r_1=0$, and $r_2=1$ are specified over the whole field as the initial

conditions of the computation to simulate the experimental conditions.

8.45 Convergence

Since there was no interfluid mass transfer between the two fluids, convergence was easier obtained in this chapter than that in Chapter 7. Only the "FALSDT" of the code was needed to relax the velocity components.

10 sweeps were required at each time step. The order of the whole-field sum of the absolute value of the volume flow residuals at the end of each time step was 10^{-9} .

Table 8.45-1 summarizes all the information about the convergence of the computations.

Variable	Value
Grid	25 grid-nodes in Y-direction 32 grid-nodes in Z-direction
Type of grid spacing	non-uniform; finer near the axis and inlet
Time-step	equal, 0.02 sec.
Number of iterative sweeps at each time-step	10
Relaxation	FALSDT: 0.1 for V_1 , V_2 0.001 for W_1 , W_2
Whole-field sum of ABS(vol. flow residuals)	1.626×10^{-9}

Table 8.45-1 Computational details for the kerosene-into-water transient jet

8.46 Computer storage and computer time

Table 8.46-1 displays the essential incore space and the computer time for different numbers of grid-nodes used for the whole domain in the computations.

Grid NYXNZ	Computer storage	Computer time per time-step (sec.)
20x20	13,489	64.48
25x25	20,394	101.04
25x28	22,596	117.14
25x32	25,542	147.65
25x35	27,734	160.2

Table 8.46-1 Computational requirement for the
kerosene-into-water transient jet
(Computer used: Perkin Elmer 3220)

8.5 Definition of the jet-tip

The jet-tip is defined in a similar way to what was adopted in Chapter 7, i.e. the jet-tip is the axial location of the horizontal plane where the mass flow rate of the injected kerosene is equal to 1% of that at the nozzle exit.

The mass flow rate of the kerosene across each horizontal plane is calculated at each time step. This mass flow

rate M can be expressed as:

$$M = 2\pi \int_0^R W_1 \rho_1 r_1 y dy \quad (8.5-1)$$

where R is the radius of the integration domain.

8.6 Computational results of tip-penetration rate

For the present computations, the main results presented and compared with the experimental data are the penetration rates of the jet-tip, i.e. the increases with time in the height of the jet-tip. They are deduced from the field values of the computational outputs according to the definition given in Section 8.5.

The flow conditions of the four cases which have been computed are summarized in Table 8.2-1 while the experimental data were obtained in Chapter 6. The computational results of the tip-penetration rate are deduced by similar steps to those described in Section 7.5.

Figs 8.6-1 to 8.6-4 show the computational and experimental results of the tip-penetration rate for the four cases considered. In these figures, the dimensionless variables Ht/D_n and $TxWi/D_n$ are adopted again, the solid lines are the computational results while

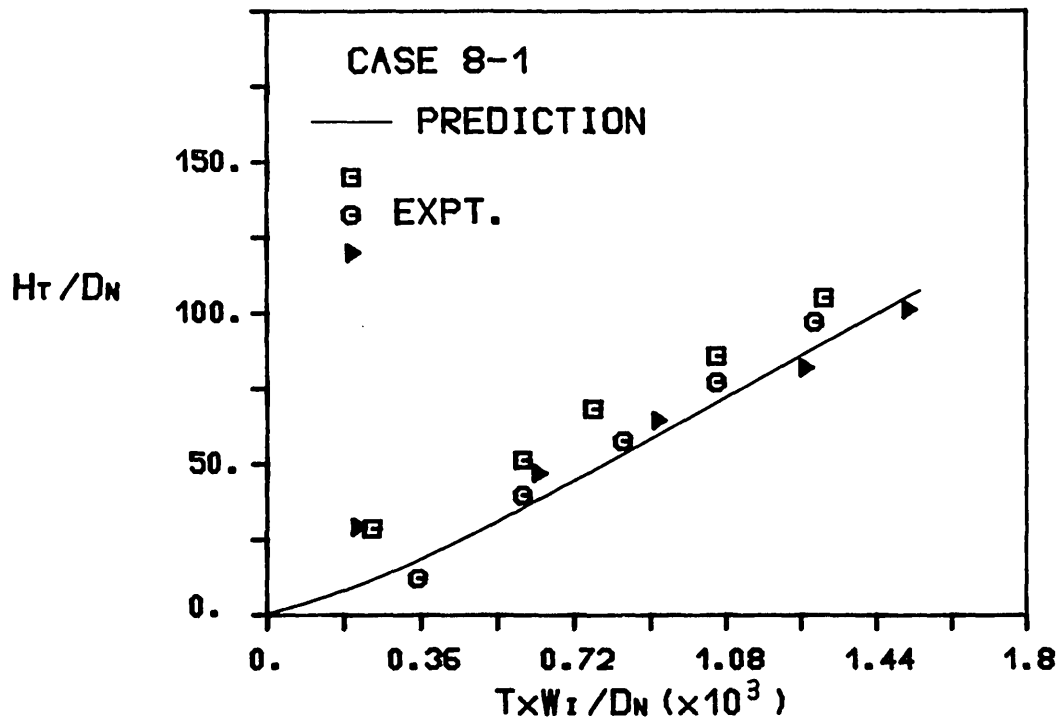


FIG. 8.6-1 TIP-PENETRATION RATE OF A KEROSENE-INTO-WATER TRANSIENT JET: CASE 8-1

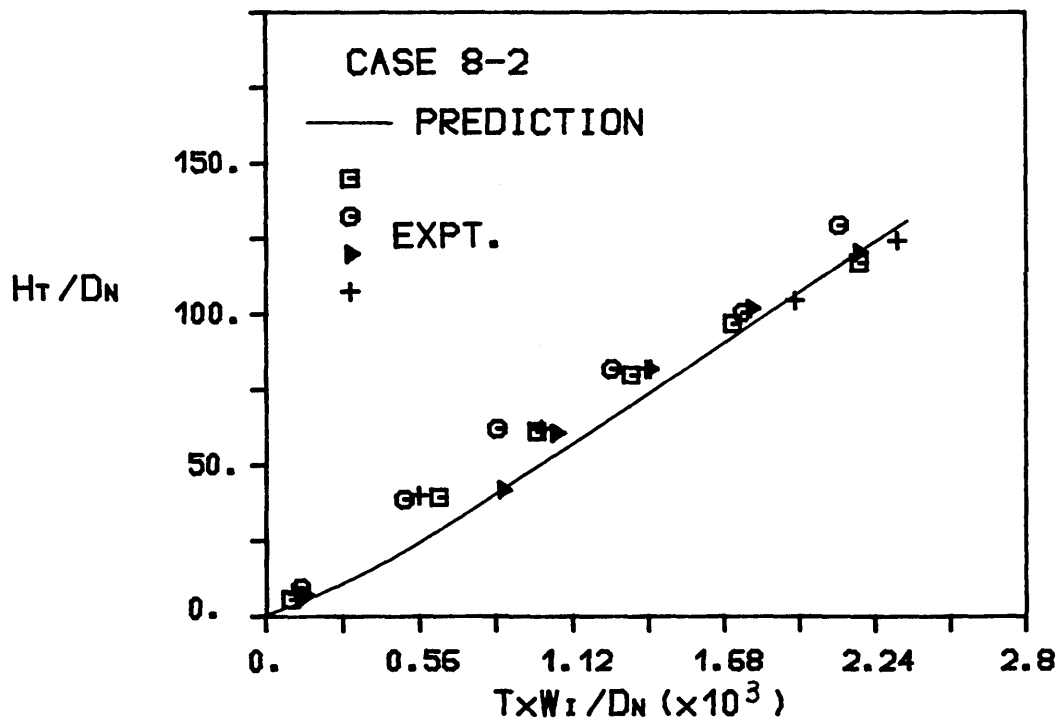


FIG. 8.6-2 TIP-PENETRATION RATE OF A KEROSENE-INTO-WATER TRANSIENT JET: CASE 8-2

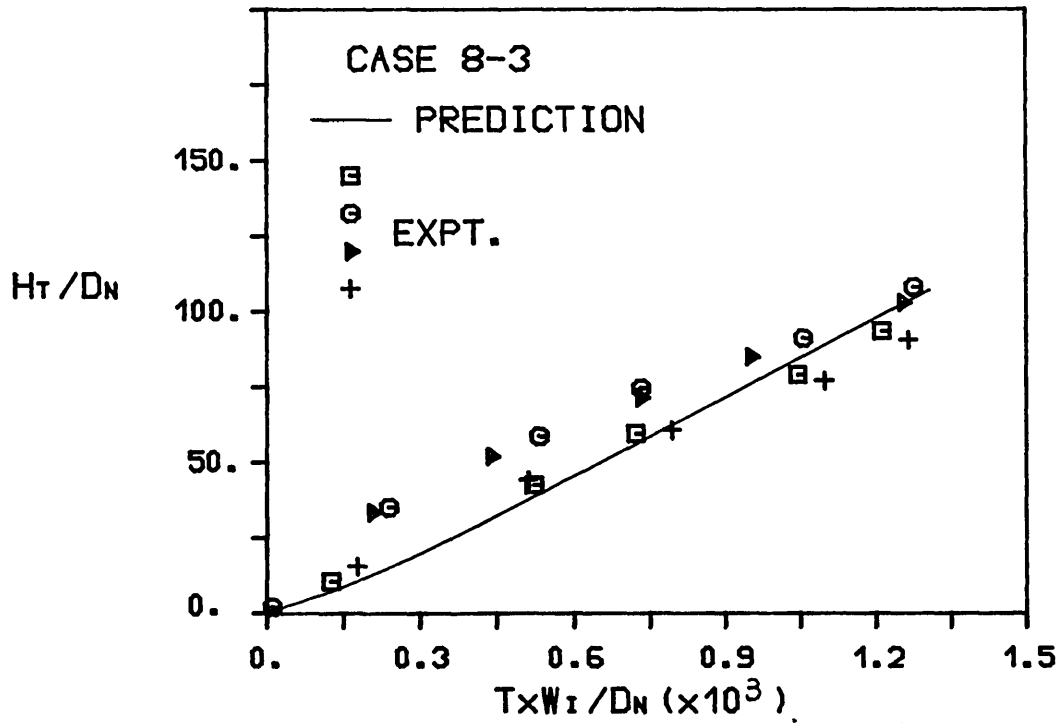


FIG. 8.6-3 TIP-PENETRATION RATE OF A KEROSENE-INTO-WATER TRANSIENT JET: CASE 8-3

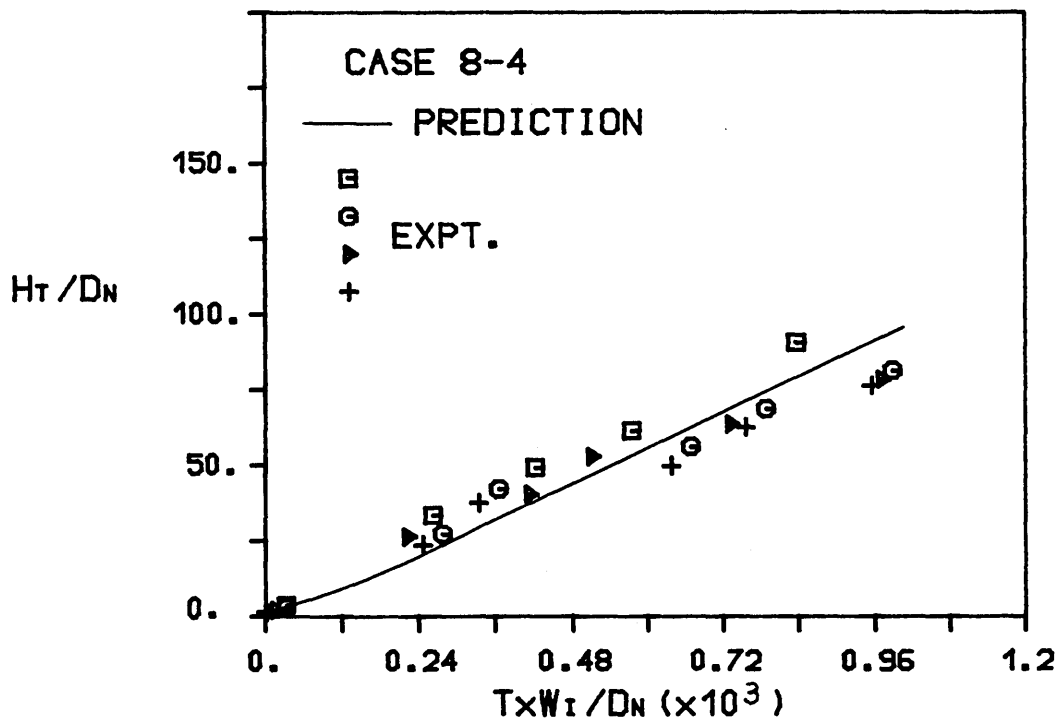


FIG. 8.6-4 TIP-PENETRATION RATE OF A KEROSENE-INTO-WATER TRANSIENT JET: CASE 8-4

the symbols are the data.

Figs. 8.6-5, 6 show the predicted profiles of the streamwise velocities and volume fraction of the kerosene at $Z/D_n=20, 30$, and 40 of Case 8-1 at $t=1.0$ sec..

8.7 Effects of empirical constant C_f and density ratio of injected fluid to surrounding water

Case 8-1 was taken as the flow to compute to demonstrate the effects of the empirical constant C_f and density ratio of the injected fluid to surrounding water. The predicted penetration rates of the jet-tip, and the predicted profiles of the streamwise velocities and volume fraction r_1 at $Z/D_n=30$ and $T=0.8$ sec., with various empirical constant C_f and density ratio are presented.

Figs. 8.7-1, 2, 3 show the effects of C_f on the predictions, while Figs. 8.7-4, 5, 6 show those of the density ratio.

8.8 Discussion

In this chapter, the kerosene-into-water transient jet has been simulated, and the predicted tip-penetration rate has been compared with the experimental data of the present study.

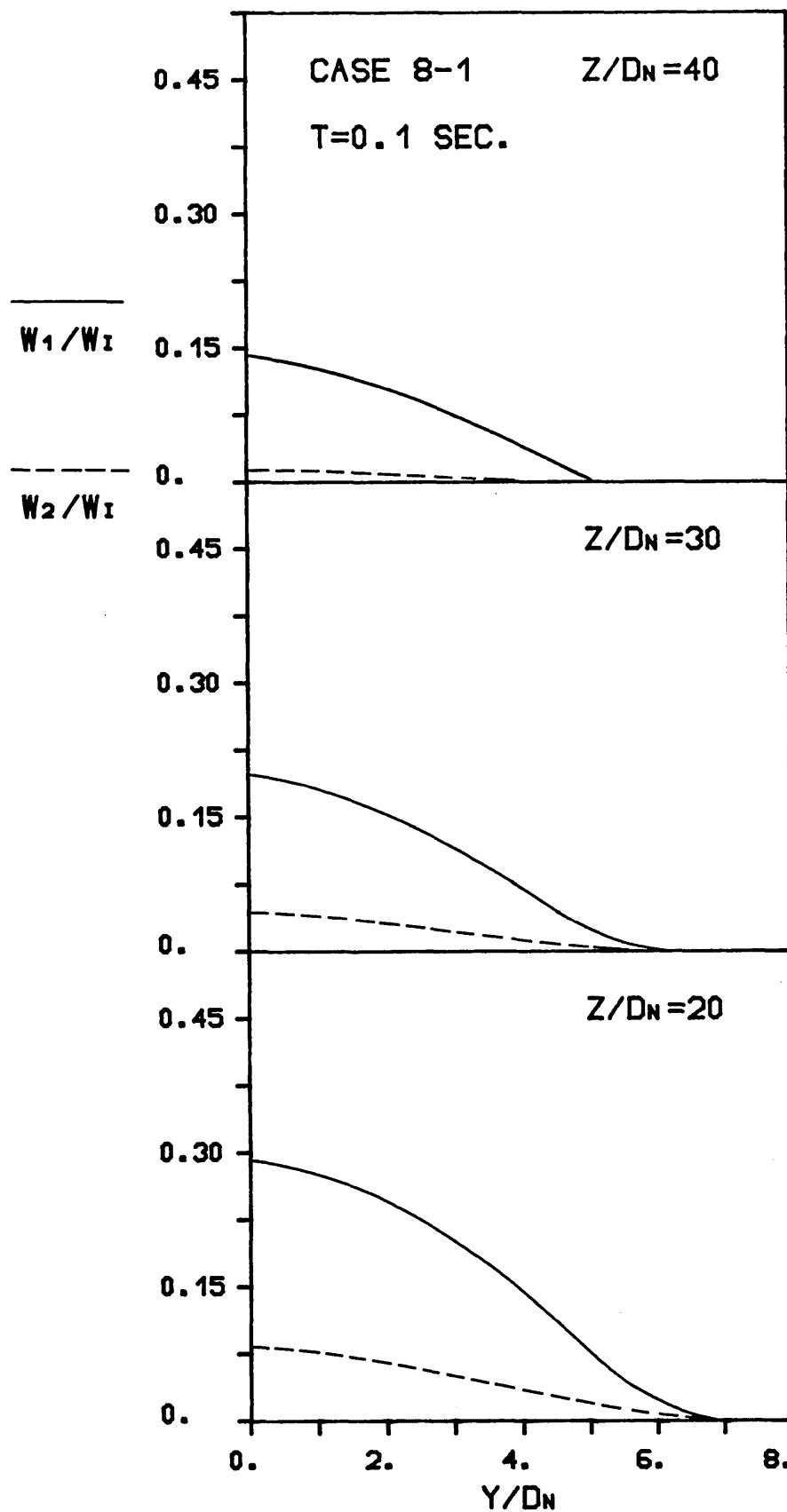


FIG. 8.6-5 PREDICTED PROFILES OF
STREAMWISE VELOCITIES

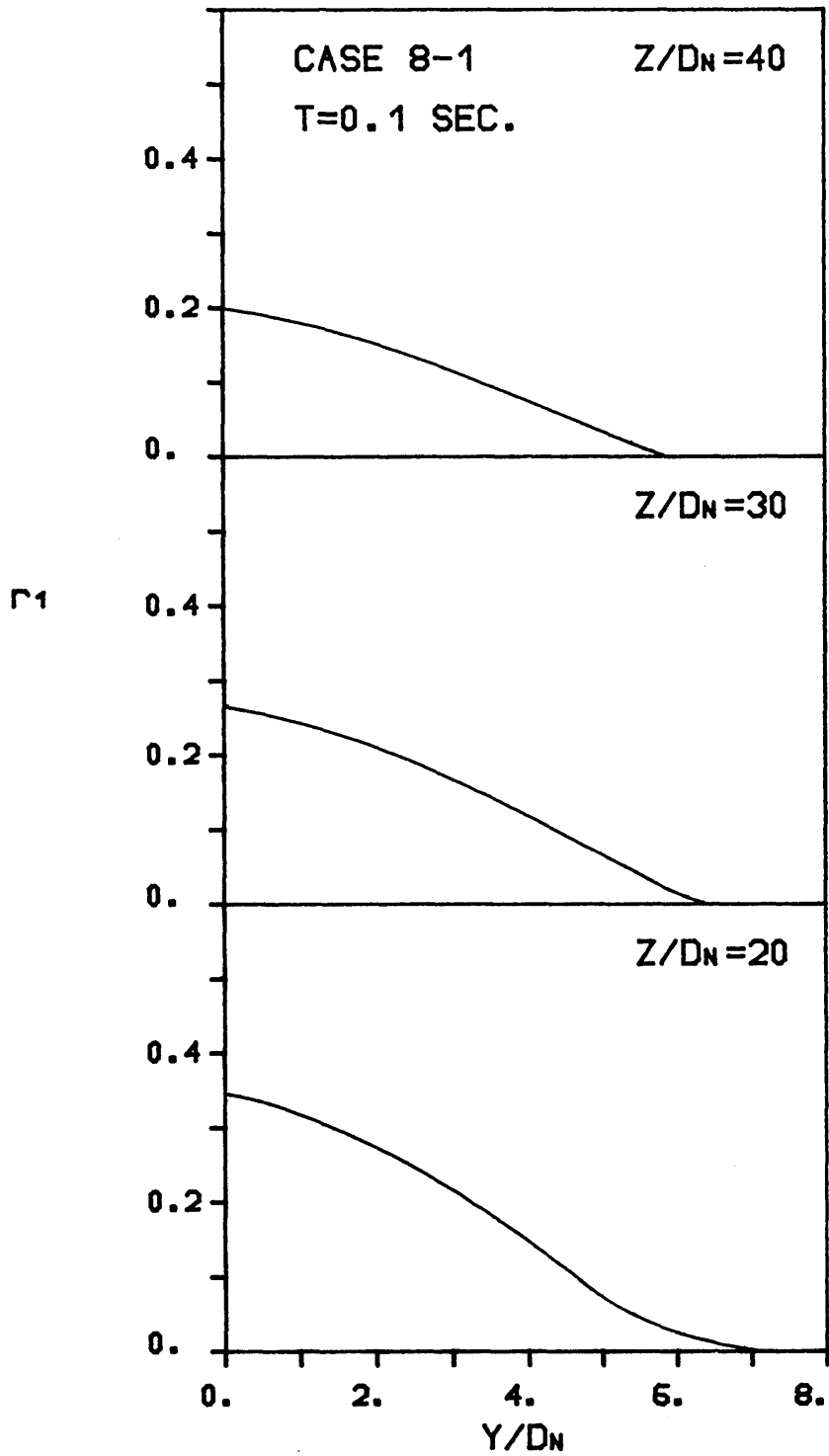


FIG. 8.6-6 PREDICTED PROFILES OF
VOLUME FRACTION OF
THE KEROSENE

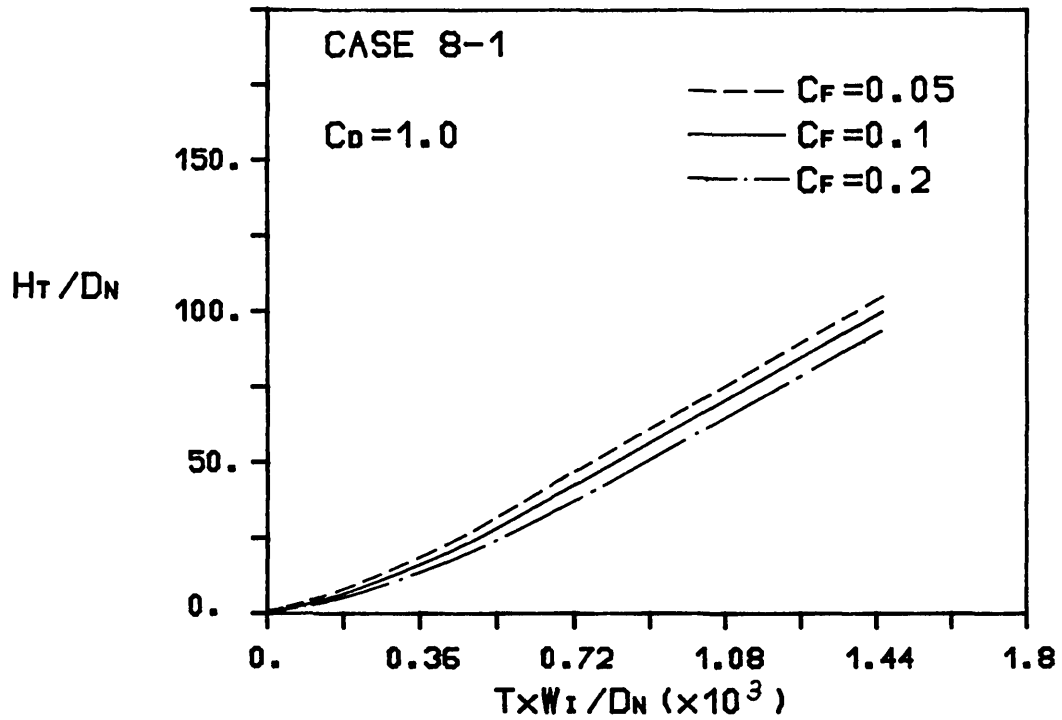


FIG. 8.7-1 EFFECT OF C_F ON THE PREDICTED TIP-PENETRATION RATE OF A KEROSENE-INTO-WATER TRANSIENT JET

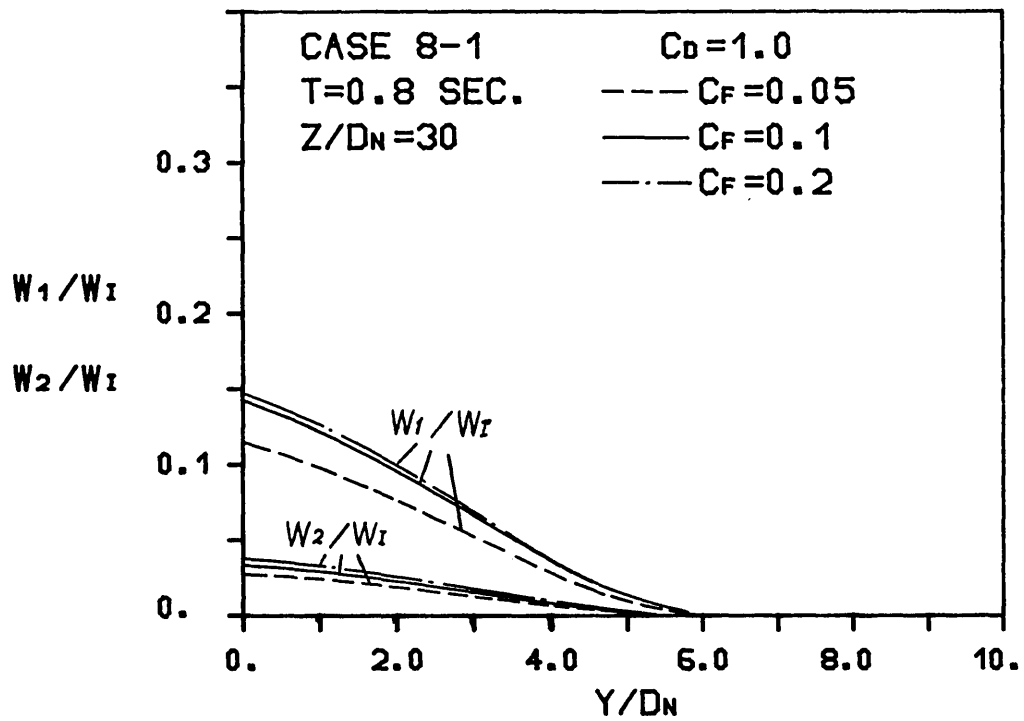


FIG. 8.7-2 EFFECT OF C_F ON THE PREDICTED PROFILES OF STREAMWISE VELOCITIES

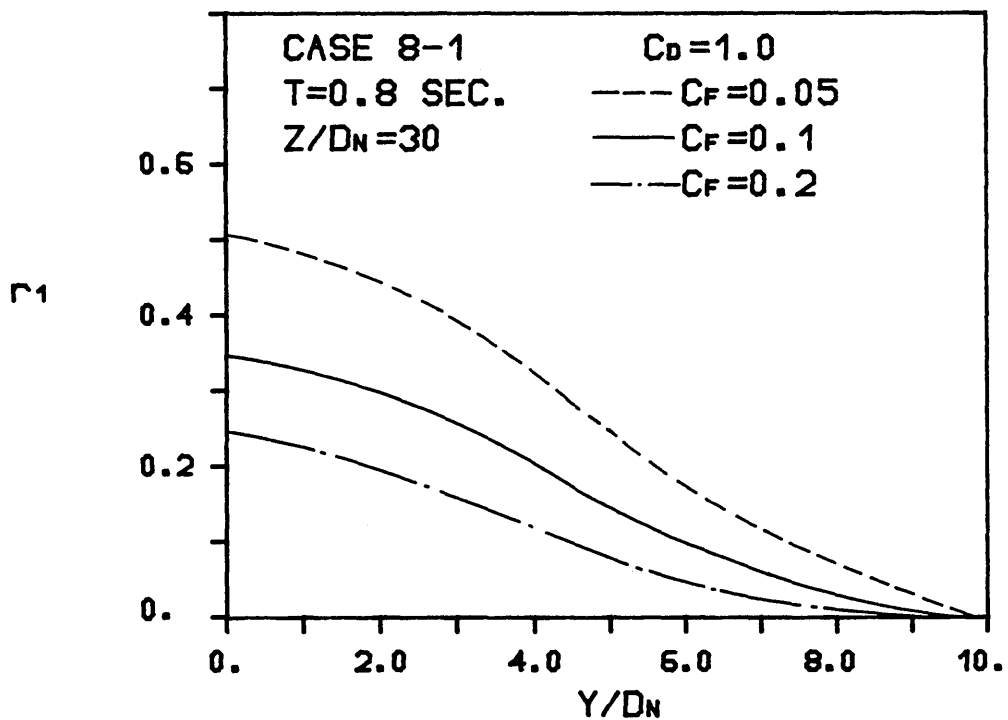


FIG. 8.7-3 EFFECT OF C_F ON THE PREDICTED PROFILE OF r_1

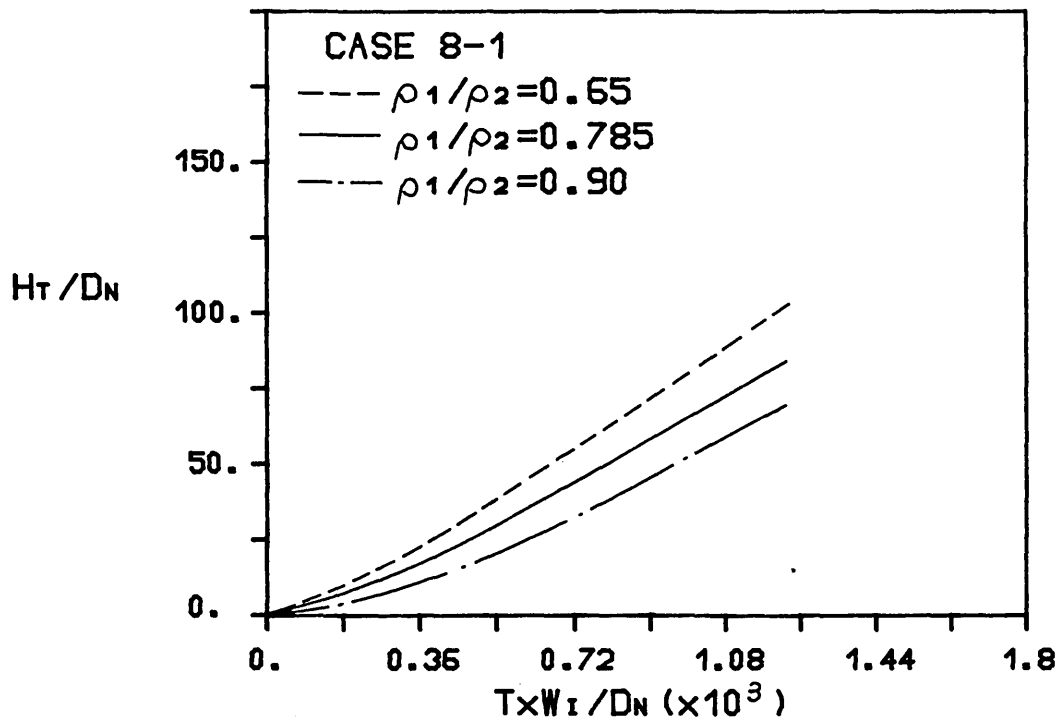


FIG. 8.7-4 EFFECT OF ρ_1/ρ_2 ON THE PREDICTED TIP-PENETRATION RATE OF A KEROSENE-INTO-WATER TRANSIENT JET

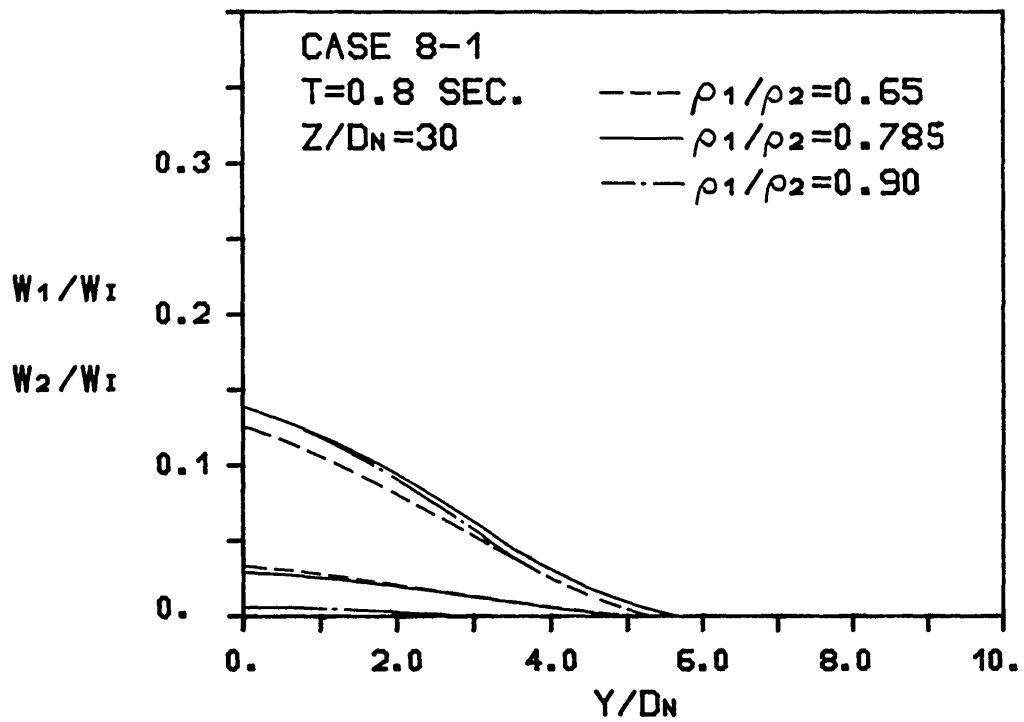


FIG. 8.7-5 EFFECT OF ρ_1/ρ_2 ON THE PREDICTED PROFILES OF STREAMWISE VELOCITIES

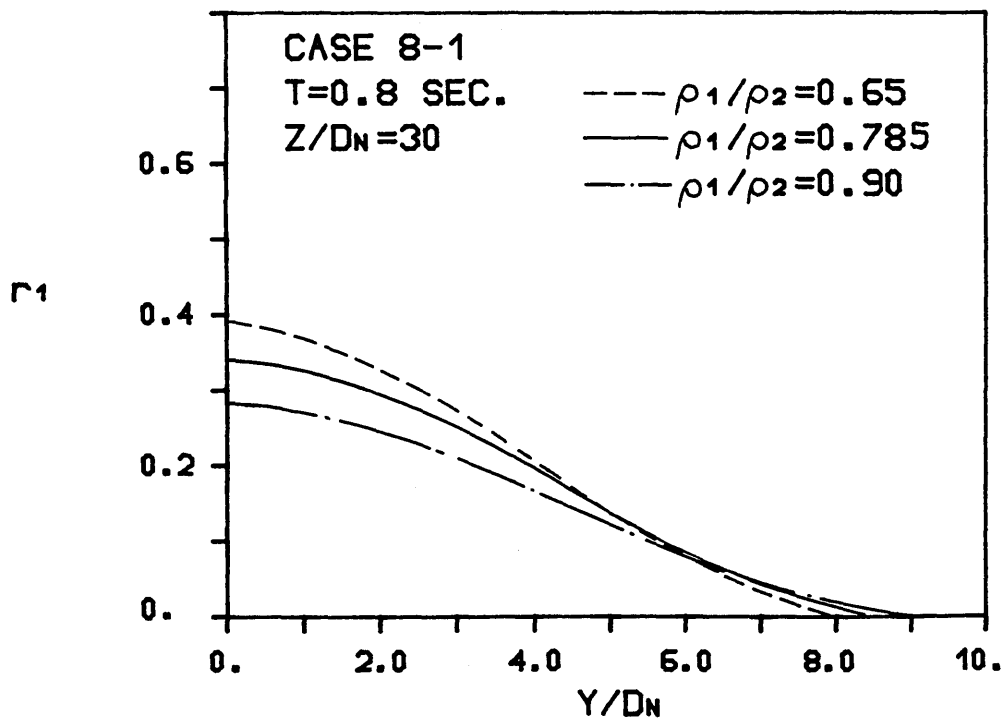


FIG. 8.7-6 EFFECT OF ρ_1/ρ_2 ON THE PREDICTED PROFILE OF Γ_1

The comparisons of the predictions with the data of the penetration rate of the jet-tip are shown in Figs. 8.6-1 to 8.6-4. From these figures, it can be found that the agreements are satisfactory in the second half of the period computed; but the computations underpredict the penetration rate in the first half of the period.

In the computations, the flow was assumed to be turbulent flow, and the constant velocity at the nozzle exit, which the injection gradually reached was taken as W_i in equation (2.41-1) to calculate the effective viscosity. The constant-effective-viscosity model thus overestimated the viscous transport of the momentum in the radial direction. This was particularly severe in the initial period, during which the nozzle exit velocity was much lower than the constant one. On the other hand, the measured mean droplet diameter within whole jet region was taken to be the mean fragment size L in the interfluid friction factor expression. In the immediate vicinity of the nozzle exit, however, the jet had not been fragmented at all, so that the interfluid friction factor should be much smaller than what was estimated by the expression. This might be another cause for the underestimation of the tip-penetration rate in the initial period.

O'Rourke and Bracco (1980) found the difficulties to reproduce the structure of a spray with any accuracy,

because the exchange of momentum between droplets and gas is functionally different to that between jets and gas. Therefore, they started their computations some 10 nozzle diameters downstream of the actual nozzle exit plane by injecting a mixture of small droplets and entrained chamber gas, with both phases moving at a same velocity. In the present study, similar difficulties have been encountered. The computations in this chapter were started from the actual nozzle exit plane; but C_f was taken a rather high value, 500, at the first axial station, in order that two phases were moving at a same velocity at that station.

No detailed tests of the empirical constants of the modelling assumptions have been performed.

The same constant-effective-viscosity model has been used to calculate the turbulent viscosity for both the miscible and immiscible flows. In the two-fluid assumptions of this chapter, however, the injected kerosene and surrounding water were defined as the two fluids, while the measured mean droplet diameter was adopted as the mean fragment size of the flow. These are different from what were used for previous miscible flows. The experiment in Chapter 6 reveals the similarities between the miscible and immiscible flows. It is, therefore, worthwhile to make efforts to develop a model which is capable of using same modelling assumptions to simulate both miscible and immiscible turbulent flows. Suggestions to develop a

model of this kind have been made by Spalding (1985b).

8.9 Conclusion

The kerosene-into-water transient jet with the experimental conditions of the present study has been simulated in this chapter.

The mathematical model used for this two-phase flow is similar to that for the single-phase flows in the previous chapters. Modifications of the two-fluid assumptions made for this flow are: (a) the two phases are defined as the two fluids; (b) the measured mean droplet diameter is taken as the mean fragment size; and (c) no interfluid mass transfer exists.

The performance of the mathematical model for this two-phase flow can be summerized as follows:

- (1) In general, the predicted tip-penetration rates agree with the experimental data satisfactorily;
- (2) In the initial period, the computations underpredict the tip-penetration rate.
- (3) The empirical constants suitable to this flow are:

$$C_{\mu} = 0.0142, \quad C_m = 0, \quad C_f = 0.1, \quad \text{and} \quad C_d = 1.0.$$

More precise measurements and tests of the modelling assumptions for the transient two-phase flow are required.

CHAPTER 9 - CONCLUSIONS

A review of previous chapters shows that all the objectives set out for the present study have been accomplished. In this conclusion chapter, the main achievements and the conclusions made in this thesis are provided below. Suggestions for future work are summarized in Section 9.2.

9.1 Achievements and conclusions

9.1.1 Computational work

Transient, turbulent, round jets with various flow conditions have been, to a certain extent, successfully numerically simulated in the present study. The flows simulated include those of miscible and immiscible fluids, with or without buoyancy, and with constant or time-dependent boundary conditions. The computations have provided the predictions for the development of the mean velocity along the centre-line, the development of the mean velocity at given axial stations, the tip-penetration rate, and the profiles of the streamwise velocities and volume fractions of two fluids.

In the mathematical model, the conservation equations governing the motion of the fluids have been stated, the constant-effective-viscosity model was used to calculate

the turbulent viscosity, while some assumptions of the two-fluid model were incorporated to preliminarily demonstrate the intermittency phenomenon of the turbulent flows.

The same constant-effective-viscosity model has been applied to both miscible and immiscible turbulent flows, while the two-fluid assumptions have been modified for the immiscible flow.

The employed modelling assumptions and the recommended empirical constants for the flows considered in this thesis can be summarized as follows: (Sections 2.4, 2.5, and 8.3 explain them in detail)

- effective viscosity, μ_{eff}

$$\mu_{eff} = C_{\mu} \rho_1 W_i D_n$$

- interfluid mass transfer, m

$$m = C_m \rho_2 r_1 r_2 |W_1 - W_2| / L$$

- interfluid friction factor, f

$$f = C_f \rho_2 r_1 r_2 |W_1 - W_2| / L$$

- gradient-diffusion coefficient, D

$$D = C_d \frac{\mu_{\text{eff}}}{\rho_1}$$

- empirical constants

for the single-phase flows:

$$C_\mu = 0.0142, \quad C_m = 0.35, \quad C_f = 0.05, \quad C_d = 1.0$$

for the two-phase flow:

$$C_\mu = 0.0142, \quad C_m = 0, \quad C_f = 0.1, \quad C_d = 1.0.$$

For the transient jet computations, non-uniformly distributed grid-nodes and expanding time-steps are preferred. These can be found in more detail, for example, in subsection 5.32.

Good agreement between the computational results and available experimental data of the air-into-air steady jet shows that the constant-effective-viscosity model with the two-fluid assumptions can predict satisfactorily the mean flow as well as the intermittency of this steady jet.

The computations have reasonably simulated the transient jets with various flow conditions. But the discrepancies between the predictions and experiments suggest, that the

refinements of the constant-effective-viscosity as well as the two-fluid assumptions are required, in order to improve the transient, turbulent flow predictions.

9.12 Experimental work

An experimental rig has been constructed in the present study. The experiments for the miscible (brine-into-water) and immiscible (kerosene-into-water) transient jets as well as the kerosene-into-water steady jet have been performed on this rig for a range of nozzle diameter and nozzle exit velocity.

The similarities between the miscible and immiscible transient, turbulent jets have been observed from the successive photographs obtained: both jets grow in a very similar manner.

The tip-penetration rates of the transient jets have been determined from a visual examination of the magnified photographic negatives. These data were used to be compared with the computational results of the present study.

The variation of the volume flow rate of the injected fluids at the nozzle exit of the transient jets has been measured. The measured data were then fitted by the Spline Function, and this has been supplied as the time-

dependent boundary condition at the nozzle exit for the computations.

For the kerosene-into-water steady jet, the droplet-size was measured. Mean droplet diameter distributions along the centre-line as well as mean droplet diameters within whole jet region have been obtained. It has been found that the mean droplet diameters in cross-sections of each jet do not change considerably along the centre-line; but those within whole jet do decrease with an increase in the *exit velocity* or a decrease in nozzle diameter.

The clear pictures of the flow patterns and useful information for the computations display that the present experiment is a simple but informative way to investigate experimentally the transient and steady jets considered in the present study.

9.2 Suggestions for future work

In connection with this study, areas in which future study may found to be profitable are listed below:

9.21 Computational

- (1) Computations of the present kind should be carried out by using two-fluid model without constant-

effective-viscosity assumption to improve the simulations of the transient jets.

- (2) Particle-tracking technique can be used together with PHOENICS to constitute a form of numerical flow visualization.
- (3) Computations of single- or two-phase round jets with chemical reaction, or other flows, such as plane jet, wake, mixing layer, should be carried out to refine the modelling assumptions of turbulent flows.
- (4) Develop a model which is capable of treating both single- and two-phase turbulent flows by using same modelling assumptions.

9.22 Experimental

- (1) The apparatus constructed for this study could be used to perform more experiments aiming at widening the range of flow conditions covered by the present study. In particular, the range of the density ratio of two fluids should be widened.
- (2) For the droplet-size measurement, various kinds of liquids other than kerosene should be used, in order to obtain more information of the droplet size in turbulent two-phase flow.

- (3) The photographic equipment with higher speed could be used in the experiment. More successive photographs within one transient run can provide more detailed flow patterns as well as more accurate data of the jet growth.

NOMENCLATURE

SYMBOLS

a	exponent, eqn. (5.32-1)
A	coefficient, eqn. (3.3-11); cell surface area, eqn. (3.3-2)
A_e	coefficient, eqn. (3.3-11)
A_n	coefficient, eqn. (3.3-11)
A_o	coefficient, eqn. (3.3-11)
A_p	coefficient, eqn. (3.3-11)
A_s	coefficient, eqn. (3.3-11)
A_w	coefficient, eqn. (3.3-11)
B	body force, table (2.3-1)
C_d	two-fluid model constant, eqn. (2.42-8)
C_f	two-fluid model constant, eqn. (2.42-7)
C_m	two-fluid model constant, eqn. (2.42-1)
C_μ	constant-effective-viscosity model constant, eqn. (2.41-1)
d	droplet diameter
d_c	characteristic diameter, eqn. (6.44-1)
$\bar{d}_j$	mean droplet diameter within whole jet
$\bar{d}_z$	mean droplet diameter in cross-section
D	"phase-diffusion" exchange coefficient, eqn. (2.42-8)
D_n	nozzle diameter
Dt	forward time-step
Dz	forward step-size in axial direction
f	interfluid friction factor, eqn. (2.42-7), or domain spread rate, eqn. (4.31-1)
F_v, F_w	friction force in y-, z-directions, table (2.3-1)
g	gravitational acceleration

Ht	height of jet-tip
I	intermittency factor
Iz	index of a grid-node
L	fragment size, eqn. (2.42-1)
L_m	mixing length, eqn. (2.42-2)
m	interfluid mass transfer rate, eqn. (2.3-1)
M	mass flow rate across horizontal plane, eqn. (7.4-1)
Ny, Nz	radial, axial grid-node number
P	pressure
r	volume fraction, eqn. (2.21-1)
R	radius of domain
Re	Reynolds number, eqn. (4.2-1)
S	source term, eqn. (2.3-3)
Su, S ϕ	source term for boundary condition, eqn. (3.3-10)
t	time
U	x-direction velocity
V	y-direction velocity, or cell volume, eqn. (3.3-2)
Vd	volume percentage of droplets, Fig. (6.44-1)
V_i	volume flow rate at nozzle exit
W	z-direction velocity
$\tilde{W}$	W/W_0
$\tilde{W}_1$	W_1/W_0
$\tilde{W}_2$	W_2/W_0
x, y, z	coordinates
Y_0	width of inlet boundary, eqn. (4.31-1)
$Y_{0.5}$	half radius of jet at which the mean velocity is half its axial value
ZWLAST	axial distance of domain, eqn. (5.32-1)

GREEK SYMBOLS

ν	kinematic viscosity
μ	viscosity
μ_{eff}	effective viscosity, eqn. (2.41-1)
ρ	density
ϕ	genegal variable
Γ	exchange coefficient, eqn. (3.3-3)
σ	interfacial tension, eqn. (6.44-1)

SUBSCRIPTS

BC	boundary condition
o	value at centre-line, or initial value of time-step
i	index for fluid-1, or -2, or condition at nozzle inlet
n	nozzle condition
ϕ	value for variable ϕ
x, y, z	x, y, z-direction components

REFERENCES

- Abramovich, S. and Solan, A (1973)
"The initial development of a submerged round jet"
J. Fluid Mech., Vol.59, P.791
- Adler, C. R., Mark, A. M., Marshall, Jr. W. R. and Parent, R. J. (1954)
"A scanning device for determining the size distribution of spray droplet images"
Chem. Eng. Progr., Vol.50, p.14
- Allano, D., Gouesbet, G., Grehan, G. and Lisiecki, D. (1982)
"Comparitive measurements of calibrated droplets using Gabor holography and corrected top-hat laser beam sizing, with discussion of simultaneous velocimetry"
International Symposium on Applications of Laser Doppler Anemometry to Fluid mechanics, Lisbon, Portugal
- Alpert, R. L. and Mathews, M. K. (1979)
"Calculation of large-scale flow field induced by droplet sprays"
Technical Report No.FMRC J.1.OEOJ4.BU, Factory Mutual Research Corp., Norwood, MA
- Ames, WF (1969)
"Numerical methods for partial differential equations"
Nelson and Sons Ltd.
- Arai, M., Shimizu, M. and Hiroyasu, H. (1985)
"Break-up length and spray angle of high speed jet"
ICLASS-85, IB/4/1
- Bhuta, P. G. (1961)
"Non-steady two-dimensional jet mixing theory for laminar and turbulent flows"
D.Sc.Diss., Stevens Inst. of Tech.
- Boulos, M.I. and Pei, D. C. (1969)

"Simultaneous heat and mass transfer from a single sphere to a turbulent air stream"
Can. J. Chem. Eng., Vol.47, p.30

Boyson, F., Ayers, W. H., Swithenbank, J. and Pan, Z. (1981)
"Three-dimensional model of spray combustion in gas turbine combustors"
AIAA Paper No.81-0324

Bracco, F. V. (1973)
"Theoretical analysis of stratified, two-phase Wankel engine combustion"
Combust. Sci. Tech., Vol.8, p.69

Bracco, F. V. (1985)
"Modeling of engine sprays"
Engine Combustion Analysis: New Approaches, 850394, Society of Automotive Engineers, Inc.

Bruce, T. W., Mongia, H. C. and Reynolds, R. S. (1979)
"Combustion design criteria validation"
Vol. I-III, USARTL-TR-78-55 (A, B, C), Fort Eustis, VA

Buckingham, A. C. (1979)
"Turbulent dusty gas motion with weak statistical coupling"
AIAA Paper No.79-1484

Buckingham, A. C. and Siekhaus, W. J. (1981)
"Interaction of moderately dense particle concentrations in turbulent flow"
AIAA Paper No.81-0346

Burstein, S. Z., Hammer, S. S. and Agosta, V. D. (1962)
"Spray combustion model with droplet breakup: analytical and experimental results"
Progress in Astronautics and Rocketry, Vol.6, P.243, Academic Press, New York

Burt, R. and Troth, K. A. (1970)
"Penetration and vaporization of Diesel fuel sprays"
Proc. I. Mech. E., Vol. 184, Part 3j, p.147

Butler, T. D., Cloutman, L. D., Dukowicz, J. K., Ramshaw, J. D. and Krieger, R. B. (1980)
"Combustion modeling in reciprocating engines"
Plenum press, New York

Chevray, R. and Tutu, N. K. (1978)
"Intermittency and preferential transport of heat in a round jet"
J. Fluid Mech., Vol.88, P.133

Chiu, W. S., Shahed, S. M. and Lyn, W. T. (1976)
"A transient spray mixing model for Diesel combustion"
SAE Trans., Vol.85, P.502

Clamen, A. and Gauvin, W. H. (1969)
"Effects of turbulence on the drag coefficients of spheres in a supercritical flow regime"
AIChE J., Vol.15, p.184

Collins, S. B. and Knudsen, J. G. (1970)
"Drop-size distributions produced by turbulent pipe flow of immiscible liquids"
AIChE J., Vol.16, No.6, p.1072

Crowe, C. T. (1974)
"A computational model for the gas-droplet flow in the vicinity of an atomizer"
Paper No.74-25, Western States Section, The Combustion Institute

Crowe, C. T. (1978)
"A Numerical model for the gas droplet flow field near an atomizer"
First International Conference on Liquid Atomization and Spray Systems, Tokyo

Dehaye, J. M., Giot, M. and Riethmuller, M. L. (1981)
"Thermohydraulics of two-phase systems for industrial
design and nuclear engineering"
Hemisphere Publishing Corp., Washington, p.95

Demetriou, J. D. (1983)
"Experiments on Starting homogeneous jets"
J. of Hydraulic Eng., Vol.109, No.7

Dent, J. C. (1971)
"A basis for the comparison of various experimental
methods for studying spray penetration"
SAE Trans., Vol.80, P.1881

Durst, F. (1982)
"Review-combined measurements of particle velocities, size
distributions and concentrations"
ASME Trans., Vol.104, P.284

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

Farmer, W. M. (1972)
"Measurement of particle size, number density and velocity
using a laser interferometer"
Applied Optics, Vol.11, P.2603

Fraser, R. P. and Eisenklam, P. (1956)
"Liquid atomization and the drop size of sprays"
Trans. Instn. Chem. Engrs., Vol.34, p.294

Galloway, T. R. and Sage, B. H. (1964)
"Thermal and material transfer in turbulent gas stream - a
method of prediction for spheres"
Int. J. Heat Mass Transfer, Vol.7, p.283

Galloway, T. R. and Sage, B. H. (1967)

"Thermal and material transfer from spheres prediction of macroscopic thermal and material transfer"
Int. J. Heat Mass Transfer, Vol.10, p.1195

Gosman, A. D. and Iderian, F. I. K. (1976)
"Teach-3p, a general computer program for two-dimensional, turbulent recirculating flows"
Dept. of Mech. Eng., Imperial College, London

Gosman, A. D. and Ioannides, E. (1981)
"Aspects of computer simulation of liquid-fueled combustors"
AIAA Paper, No.81-0323

Gosman, A. D. and Johns, R. J. R. (1980)
"Computer analysis of fuel-air mixing in direct-injection engines"
SAE paper, No.800091

Gray, J. T., Mecher, N. T. and Mannheimer, R. J. (1966)
"Some observations on the liquid injection techniques as a research tool"
SAE Trans., Vol. 75, paper 660748

Handscomb, D. C. (1965)
"Methods of numerical approximation"
Pergamon Press

Harlow, F. H. and Amsden, A. A. (1975a)
"Numerical calculation of multiphase fluid flow"
J. Comp. Phys., Vol.17, p.19

Harlow, F. H. and Amsden, A. A. (1975b)
"Flow of interpenetrating material phases"
J. Comp. Phys. Vol.18, p.440

Harlow, F. H. and Welch, J. E. (1965)
"Numerical calculation of time-dependent viscous incompressible flow of fluid with a free surface"

The Physics of Fluids, Vol.8, No.12

Hay, N. and Johnes, P. L. (1972)
"Comparison of the various correlations for spray
penetration"
SAE Paper No. 720776

Hayworth, C. B. and Treybal, R. E. (1950)
"Drop formation in two-liquid-phase systems"
Industrial and Engineering Chemistry, Vol.42, No.6

Hengrussamee, D. (1975)
"Mixing of hot starting-jets with cold surrounding"
Ph.D. Thesis, University of London

Hiroyasu, H. and Kadota, T. (1974)
"Droplet size distributions in Diesel engines"
SAE Trans., Vol.83, Paper No.740715, p.2615

Hiroyasu, H. and Kadota, T. (1976)
"Models for combustion and formation of nitric oxide and
soot in direct injection Diesel engines"
SAE Trans. Vol.85, p.513

Hiroyasu, H., Toyota, Y. and Kadota, T. (1978)
"Transient characteristics of droplet size distribution in
Diesel sprays"
Paper No.5-3, presented at the First International
Conference on Liquid Atomization and Spray Systems, Tokyo

Hishida, K., Maeda, M., Imasu, J., Hirogana, K. and Kano,
H. (1982)
"Measurements of size and velocity of particle in two-
phase flow by a three beam LDA system"
International Symposium on Application of Laser Doppler
Anemometry to Fluid Mechanics, Lisbon, Portugal

Holfelder, O. (1932)
"Der Einspritzvorgang bei Dieselmotoren"

Z. V. D. I., Vol.76, No.51

Ilegbusi, J. O. (1984)

"Progress in the development of a two-fluid model of turbulence for near-wall flows"

CFD/84/7, Imperial College, London

Ilegbusi, J. O. (1985)

"Calculation of turbulent flows in conduits and free shear layers with a two-fluid model of turbulence"

To be published

Ishii, M. and Chawla, T. C. (1979)

"Local drag laws in dispersed two-phase flow"

NUREG/CR-1230, ANL-79-105

Jones, A. R. (1979)

"Scattering of electro-magnetic radiation in particle laden fluids"

Prog. Energy Combustion Sci., Vol.5, p.73

Jurewicz, J. T. and Stock, D. E. (1976)

"A numerical model for turbulent diffusion in gas-particle flows"

ASME, Paper No.76-WA/FE-33

Jurewicz, J. T., Stock, D. T. and Crowe, C. T. (1977)

"The effect of turbulent diffusion on gas particle flow in an electric field"

First Symposium on Turbulent Shear Flows, p.12.27

Kent, J. C. (1972)

"Unsteady viscous jet flow into stationary surroundings"

Computers and Fluids, Vol.1, p.101

Kerker, M. (1969)

"The scattering of light"

Academic Press, New York

Khalil, E. E. and Whitelaw, J. H. (1976)
"Aerodynamic and thermodynamic characteristics of
kerosene-spray flames"
Sixteen Symposium (International) on Combustion, p.569

Khalil, E. E. (1978)
"A simplified approach for the calculation of free and
confined spray flames"
AIAA, Paper No.78-029

Kinter, R. C., Horton, T. J., Graumann, R. E. and
Amberkar, S. (1961)
"Photography in bubble and drop research"
Can. J. Chem. Eng., Vol.39, p.235

Kuo, T.-W (1982)
"On the scaling of transient laminar, turbulent, and spray
jets"
Ph.D. Thesis, Princeton Univ.

Kuo, T.-W and Bracco, F. V. (1982)
"On the scaling of impulsively started incompressible
turbulent round jets"
J. of Fluid Eng., Vol.104, p.191

Launder, B. E., Morse, A., Rodi, W. and Spalding, D. B.
(1972)
"The prediction of free shear flows-a comparison of the
performance of six turbulence models"
Paper presented at the NASA Conference on Free Shear
Flows, Langley, Hampton, Virginia

Launder, B. E. and Spalding, D. B. (1973)
"The numerical computations of turbulent flows"
Report HTS/73/2, Imperial College

Lavender, W. J. and Pei, D. C. T. (1967)
"The effect of fluid turbulence on the rate of heat
transfer from spheres"
Int. J. Heat Mass Transfer, Vol.10, p.529

Libby, P. A. (1975)

"On the prediction of intermittent turbulent flows"
J. Fluid Mech., Vol.68, p.273

Lyshevskiy, A. S. (1955)

"Determination of the length of an atomized fuel spray"
Trudy Novocherkasskogo, Politeknicheskogo Instituta,
Vol.26, p.391

Lyshevskiy, A. S. (1956)

"The coefficient of free turbulence in a jet of atomized
liquid fuel"
NASA TT-F351

Ma, A. S. C. and Ong, K. S. (1971)

"Impulsive-injection of air into stagnant surroundings"
Paper presented at IUTAM symposium on the Recent Research
on Unsteady Boundary Layer, p.1952

Malin, M. R. (1983a)

"Progress in the development of an intermittency model of
turbulence"
CFD/83/7, Imperial College, London

Malin, M. R. (1983b)

"Calculations of intermittency in self-preserving, free
turbulent jets and wakes"
CFD/83/10, Imperial College, London

Malin, M. R. and Spalding, D. B. (1984)

"A two-fluid model of turbulence and its application to
heated plane jets and wakes"
CFD/84/19, Imperial College, London

Mao, C-P., Wakamatsu, Y. and Faeth, G. M. (1981)

"A simplified model of high pressure spray combustion"
Eighteenth Symposium (International) on Combustion, p.337

McCreath, C., Roett, M. and Chigier, N. (1972)

"A technique for measurement of velocities and size of particles in flames"
J. Phys., Vol.E5, p.601

Mehlig, H. (1934)
"Zur Physik der Brennstoffstrahlen in Dieselmotoren"
A. T. Z., Vol.37, No.16

Meyer, A. P. (1961)
"An experimental investigation of the impulsive start of a 2-dimensional jet"
M.Sc. Thesis, Stevens Institute

Mongia, H. C. and Smith, K. (1978)
"An empirical/analytical design methodology for gas turbine combustors"
AIAA, Paper No.78-998

Newman, J. A. and Brzustowski, T. A. (1971)
"Behaviour of a liquid jet near the thermodynamic critical region"
AIAA J., Vol.9, p.1595

Ogasawara, M. and Sami, H. (1967)
"Study on the behavior of a fuel droplet injected into the combustion chamber of a Diesel engine"
SAE Trans., Vol.78, Paper No.670468, p.1690

Ong, K. S. (1970)
"Impulsive-injection of air into stagnant surroundings"
Ph.D. Thesis, University of London

O'Rourke, P. J. and Bracco, F. V. (1980)
"Modeling of drop interactions in thick sprays and a comparison with experiments"
Stratified Charge Automotive Engines Conference, The Institute of Mechanical Engineers, London

Öz, I. H. (1969)

"Calculation of spray penetration in Diesel engines"
SAE Trans., Vol.78, Paper No.690254, p.1107

Parks, M., Polonski, C. and Toye, R. (1966)
"Penetration of Diesel fuel sprays in gases"
SAE Paper No.660747

Patankar, S. V. and Spalding, D. B. (1972)
"A calculation procedure for heat mass and momentum
transfer in three-dimensional parabolic flows"
Int. J. Heat Mass Transfer, Vol.15, p1787

Payne, R. B. (1958)
"A numerical method for calculating the starting and
perturbation of a two-dimensional jet at low Reynolds
number"
Aeronautical Research Council Report and Memoranda No.3047

Peskin, R. L. and Kau, C. J. (1979)
"Numerical simulation of particulate motion in turbulent
gas-solid channel flow"
J. Fluids Engr., Vol.101, p.319

Phillips, H. (1971)
"Theory of suppression of explosions by narrow gaps"
Fourth Symposium on chemical process hazards with special
reference to plant design, p.17

Priem, R. J. and Heidmann, M. F. (1960)
"Propellant vaporization as a design criteria for rocket
engine combustion chambers"
NASA TR R-67

Raithby, G. D. and Eckert, E. R. G. (1968)
"The effect of turbulence parameters and support position
on the heat transfer from spheres"
Int. J. Heat mass Transfer, Vol.11, p.1233

Rayleigh, Lord (1878)

"On the instability of jets"
Mathematical Society Proc. (London), Vol.10

Rayleigh, Lord (1892)
"On the instability of a cylinder of viscous liquid under capillary force"
Phil. Mag, and J. Sci. (London)

Reitz, R. D. (1978)
"Atomization and other breakup regimes of a liquid jet"
Ph.D. Thesis, Princeton University

Rife, J. and Heywood, J. B. (1974)
"Photographic and performance studies of Diesel combustion with a rapid compression machine"
SAE Trans., Vol.83, Paper No.740948, p.2942

Rizk, W. (1955)
"Study of flow in an impulsive jet"
M. E. R. L., (East Kilbride), Fluids Report No. 28

Rodi, W. (1972)
"The prediction of free turbulent boundary layers by use of a two-equation model of turbulence"
PH.D. Thesis, University of London

Rose, J. W. and Cooper, J. R. (1977)
"Technical data of fuel"
Scottish Academic Press

Rosten, H. I. and Spalding, D. B. (1984)
"PHOENICS-84 reference handbook"
CHAM UK/TR/100, CHAM LTD, London

Ruscello, L. V. and Hirleman, E. D. (1981)
"Determining droplet size distributions of sprays with a photodiode array"
WSS/CI-81-49

Rusinov, R. V. (1963)
"Length of atomized fuel jet in a Diesel engine"
Russian Engrg. Jrn., Vol.43

Saito, A., Ohsawa, K., Kawamura, K., Shirakawa, S. and
Tanasawa, Y.(1985)
"Spray characteristics of swirl nozzles used for Diesel
engines"
ICLASS-85, IIB/3/1

Schlichting, H. (1966)
"Boundary layer theory"
McGraw-hill Book Company, 6-th edition

Schweitzer, P. H. (1937)
"Penetration of oil sprays"
Pennsylvania State Univ. Bulletin, No.46

Shahed, S. M., Flynn, P. F. and Lyn, W. T. (1980)
"Combustion modeling in reciprocating engines"
Plenum Press, New York

Shearer, A. J. and Faeth, G. M. (1977)
"Combustion of liquid sprays at high pressure"
NASA Cr-135210

Simmons, H. C. (1977)
"The correlation of drop-size distribution in fuel nozzle
sprays"
ASME Trans., J. of Engineering for Power, p.309

Sitkei, G. (1964)
"Kraftstoffaufbereitung und Verbrennung bei Dieselmotoren"
Springer-Verlag

Soo, S. L. (1967)
"Fluid dynamics of multiphase systems"
Blaisdell, Waltham, MA

Spalding, D. B. (1959)
"A one-dimensional theory of liquid fuel rocket combustion"
A. R. C. Tech, Rept., No.20-175, Current Paper No.445

Spalding, D. B. (1972)
"A novel finite-difference formulation for differential equations involving first and second derivatives"
Int. J. Num. Mech. In Eng. Vol.4

Spalding, D. B. (1975)
"GENMIX: a general computer program for two-dimensional parabolic phenomena"
HTS/75/17, Imperial college, London

Spalding, D. B. (1979)
"Combustion and mass transfer"
Pergamon Press

Spalding, D. B. (1981)
"A general purpose computer program for multi-dimensional one- and two-phase flows"
Mathematics and Computers in Simulation, North Holland Press, Vol.XXIII, p.267

Spalding, D. B. (1982a)
"Chemical reaction in turbulent fluids"
CFD/82/8, Imperial College, London

Spalding, D. B. (1982b)
"Four lectures on the PHOENICS computer code"
CFD/82/5, Imperial College, London

Spalding, D. B. (1984a)
"The two-fluid model of turbulence applied to combustion phenomena"
Paper contributed to 22-nd AIAA Meeting, Reno, Nevada

Spalding, D. B. (1984b)
"Notes on the mathematical basis of PHOENICS-84"
Technical Report Number 103, CHAM LTD, London

Spalding, D. B. (1985a)
"A turbulence model of buoyant and combusting flows"
CFD/85/10, Imperial College, London

Spalding, D. B. (1985b)
"Computational fluid dynamics and its application to
liquid-atomisation and spray systems"
Keynote Lecture at ICLASS

Sutton, R. D. (1970)
"Propellant spray combustion process during stable and
unstable liquid rocket combustion"
AFOSR TR 70-2714, Washington, D. C.

Swithenbank, J., Beer, J. M. Taylor, D. C., Abbot, D. and
McCreath, G. C. (1976)
"A laser diagnostic technique for the measurement of
droplet and particle size distribution"
AIAA Paper No.76-69, AIAA 14-th Aerospace Sciences
Meeting, Washington, D. C.

Swithenbank, J., Turan, A. and Felton, P. G. (1980)
"Gas turbine combustor design problems"
Hemisphere Publishing, Washington

Takeuchi, K., Murayama, H., Senda, J. and Yamada, K.
(1983)
"Droplet size distribution in Diesel fuel sprays"
Bulletin of the JSME, Vol.26, p.797

Taylor, D. H. (1967)
"The analysis of fuel spray penetration and distribution
in a medium speed Diesel engine using optical techniques"
Ph.D. Thesis, Loughborough University of Technology

Taylor, D. H. and Walsham, B. E. (1970)
"Combustion processes in a medium speed Diesel engine"
Proc. I. Mech. E., Vol.184, Part3J, p.67

Thring, M. W. and Newby, M. P. (1953)
"Combustion length of enclosed turbulent jet flames"
Fourth Symposium (International) on Combustion, p.789

Tishkoff, J. M., Hammond, D. C. Jr. and Chraplyvy, A. R.
(1980)
"Diagnostic measurements of fuel spray dispersion"
ASME Paper No. 80-WA/HT-35

Torobin, L. B. and Gruvin, W. H. (1961)
"The drag coefficients of single spheres moving in steady
and accelerated motion in a turbulent fluid"
AIChE J., Vol.7, p.615

Wakuri, Y., Fujii, M., Amitani, T. and Tsuneya, R. (1960)
"Studies of the penetration of a fuel spray in a Diesel
engine"
Bull. JSME, Vol.3, No.9, p.123

Wallis, G. B. (1969)
"One-dimensional two-phase flow"
McGraw-hill Book Company

Weber, C. (1931)
"Zum Zerfall eines Flüssigkeitsstrahles"
Z. A. M. M., Vol.2, No.4

Westbrook, C. K. (1977)
"Three dimensional numerical modeling of liquid fuel"
Sixteenth Symposium (International) on Combustion, P1517

Williams, F. A. (1965)
"Combustion theory"
Addison-Wesley Publishing Company

Wirz, H. J. (1973)
"Similarity solution of unsteady, compressible plane and
axisymmetric laminar boundary layer equations"
A Report Published By the University of Berlin

Witze, P. O. (1980)
"The impulsively started incompressible turbulent jet"
SAND 80-8617

Wolfhard, H. G. and Bruszk, A. E. (1959)
"Communication of flame through cylindrical channels"
U.S. Bureau of Mines Report of Investigation No. 5457

Wu, K.-J., Reitz, R. D. and Bracco, F. V. (1985)
"Drop sizes of atomizing jets"
Submitted for publication

Wynanski, I. and Fiedler, H. (1969)
"Some measurements in the self-preserving jet"
J. Fluid Mech., Vol.38, p.577

Yule, A. J., Mo, S.-L., Tham, S. Y. and Aval, S. M. (1985)
"Diesel spray structure"
ICLASS-85, IIB/2/1

APPENDIX A THE PHOENICS LISTINGS FOR THE AIR-INTO-AIR
TRANSIENT JET

A.1 "Q1"

```
*****
*                                     *
*          AIR-INTO-AIR              *
*          TRANSIENT JET            *
*                                     *
*          XI SHI-TONG              *
*          IMPERIAL COLLEGE/CFDU     *
*          1985                     *
*                                     *
*****
```

GROUP 1. RUN TITLE & PHUNNI ACCOUNT NUMBER
TEXT(AIR-INTO-AIR TRANSIENT JET

GROUP 2. TRANSIENCE, TIME-STEP SPECIFICATION
STEADY=.F.
LSTEP=100
TLAST=GRND

GROUP 3. X-DIRECTION GRID SPECIFICATION
CARTES=.F.

GROUP 4. Y-DIRECTION GRID SPECIFICATION
YVLAST=0.635
NY=25
YFRAC(1)=-5.
YFRAC(2)=0.004
YFRAC(3)=10.
YFRAC(4)=0.028
YFRAC(5)=10.
YFRAC(6)=0.07

GROUP 5. Z-DIRECTION GRID SPECIFICATION
GRDPWR(Z,32,2.54,1.5)

GROUP 7. VARIABLES STORED, SOLVED & NAMED
ONEPHS=.FALSE.
SOLUTN(P1,Y,Y,N,N,N,Y)
SOLUTN(V1,Y,Y,N,N,N,Y)
SOLUTN(V2,Y,Y,N,N,N,Y)
SOLUTN(W1,Y,Y,N,N,N,Y)
SOLUTN(W2,Y,Y,N,N,N,Y)
SOLUTN(R1,Y,Y,N,N,N,Y)
SOLUTN(R2,Y,Y,N,N,N,Y)
INTMDT=20
NAME(20)=MDOT
SOLUTN(20,Y,N,N,N,N,N)
NAME(21)=W
SOLUTN(21,Y,N,N,N,N,N)

GROUP 9. PROPERTIES OF THE MEDIUM (OR MEDIA)
*** EFFECTIVE VISCOSITY MODEL ENUT=0.02845*W0*R0
ENUT=0.044

GROUP 10. INTER-PHASE-TRANSFER PROCESSES AND PROPERTIES
CFIPS=GRND
CMDOT=GRND

GROUP 11. INITIALIZATION OF VARIABLE OR POROSITY FIELDS

```
FIINIT(R1)=1.E-10
FIINIT(R2)=1.
  GROUP 13. BOUNDARY CONDITIONS AND SPECIAL SOURCES
    JET EXIT
    PATCH(1,LOW,1,1,1,5,1,1,1,1000)
    COVAL(1,P1,FIXFLU,121.92)
    COVAL(1,W1,ONLYMS,121.92)
    WALL
    PATCH(2,LWALL,1,1,6,25,1,1,1,1000)
    COVAL(2,V1,1,0,0,0)
    COVAL(2,V2,1,0,0,0)
    COVAL(2,W1,1,0,0,0)
    COVAL(2,W2,1,0,0,0)
    FREE STREAM
    PATCH(3,NORTH,1,1,25,25,1,32,1,1000)
    COVAL(3,P2,FXP,0,0)
    COVAL(3,W2,ONLYMS,0,0)
    COVAL(3,W1,ONLYMS,0,0)
    OUTLET OF JET
    PATCH(4,CELL,1,1,1,25,32,32,1,1000)
    COVAL(4,P1,FXP,0,0)
    COVAL(4,P2,FXP,0,0)
  GROUP 15. TERMINATION OF SWEEPS
    LSWEET=15
    RESREF(P1)=1.E-4
    RESREF(V1)=1.E-4
    RESREF(V2)=1.E-4
    RESREF(W1)=1.E-4
    RESREF(W2)=1.E-4
    RESREF(R1)=1.E-4
    RESREF(R2)=1.E-4
  GROUP 17. UNDER-RELAXATION DEVICES
    RELAX(R1,LINRLX,0.1)
    RELAX(R2,LINRLX,0.1)
    RELAX(W1,FALSDT,0.1)
    RELAX(W2,FALSDT,0.1)
    RELAX(V1,FALSDT,0.1)
    RELAX(V2,FALSDT,0.1)
  GROUP 19. EARTH CALLS TO GROUND SUBROUTINE
    *** RG(1) IS CF
    RG(1)=0.05
    *** RG(2) IS CM
    RG(2)=0.35
    *** RG(3) IS THE DIA. OF THE NOZZLE
    RG(3)=0.0254
    *** RG(4) .INITIAL TIME STEP
    RG(4)=1.E-5
    *** RG(5) .EXP. COEFF. OF DT
    RG(5)=1.1
    *** DT < ,= RG(6)
    RG(6)=1.E-3
  GROUP 22. MONITOR PRINT-OUT
    NPRMON=10
    NPRMNT=10
    ITABL=0
    IPLTL=25
    NUMCLS=10
    NCOLCO=10
  GROUP 23. FIELD PRINT-OUT & PLOT CONTROL
    NTPRIN=20
```

```
OUTPUT(P1,Y,Y,Y,Y,Y,Y)
OUTPUT(W1,Y,Y,Y,Y,Y,Y)
OUTPUT(W2,Y,Y,Y,Y,Y,Y)
OUTPUT(R1,Y,Y,Y,Y,Y,Y)
OUTPUT(20,Y,Y,Y,Y,Y,Y)
OUTPUT(21,Y,Y,Y,Y,Y,Y)
TSTSMP=10
STOP
```

A.2 "GROUND"

```
$BATCH
C$DIR**MAIN
C FILE NAME GROUND.FTN-----170785
C
C   PROGRAM MAIN
C
C   GROUND.FTN TO BE USED IN CONJUNCTION WITH THE Q FILE
C   WHICH SOLVES THE PROBLEM OF AIR-INTO-AIR TRANSIENT
C   JET
C
C   XI SHI-TONG
C   IMPERIAL COLLEGE/CFDU/1985
C
C*****
C      SUBROUTINE GROUND
C$INCLUDE SATEAR/G
C$INCLUDE GRDLOC/G
C$INCLUDE GRDEAR/G
CXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX USER SECTION STARTS:
C
C 1   Set dimensions of satellite-to-GROUND data arrays to those
C     of the satellite.
C     COMMON/LGRND/LG(10)/IGRND/IG(10)/RGRND/RG(10)/CGRND/CG(10)
C     LOGICAL LG,USEGRD
C     CHARACTER*4 CG
C
C 2   User dimensions own arrays here, for example:
C     DIMENSION UUH(10,10),UUC(10,10),UUX(10,10),UUZ(10)
C----- 2-D ARRAYS
C     DIMENSION UUV1(25,1),UUV2(25,1),UUV1(25,1),UUV2(25,1)
C     DIMENSION UUR1(25,1),UUR2(25,1),UUV(25,1),UUVOL(25,1)
C     DIMENSION UUADM(25,1),UUFIP(25,1),UUMDT(25,1)
C
C----- 1-D ARRAYS AT Z-DIRECTION
C
C     DIMENSION UUL(32),UUZWNZ(32)
C
C*****
C
C--- GROUP 1. Run title
C
C     1 CONTINUE
C     CALL MAKE(VOL)
C     RETURN
C*****
C
C--- GROUP 2. Transience; time-step specification
C
C     2 CONTINUE
C     IF(ISTEP.NE.1) GO TO 201
C     DT=RG(4)
C     GO TO 202
C 201 DT=DT*RG(5)
```

```
202 DT=AMIN1(DT,RG(6))
RETURN
C*****
C
C--- GROUP 10. Inter-phase-transfer processes and properties
C
  10 GO TO (101,102,103,104),ISC
  101 CONTINUE
C--- for CFIPS,LE,GRND--- Inter-phase friction coefficient (FIP).
C----- GET VARIABLE NEEDED: V1,V2,W1,W2,R1,R2,VOL
C
  CALL GETYX(W1,UW1,NY,NX)
  CALL GETYX(W2,UW2,NY,NX)
  CALL GETYX(R1,UUR1,NY,NX)
  CALL GETYX(R2,UUR2,NY,NX)
  CALL GETYX(VOL,UVOL,NY,NX)
C
  DO 1011 IY=1,NY
  DO 1011 IX=1,NX
C
C----- CALCULATE ADM,ABSOLUTE W-VELOCITY DIFFERENCE BETWEEN FLUIDS
C
  UUADM(IY,IX)=ABS(UW2(IY,IX)-UW1(IY,IX))
C
C----- CALCULATE FIP
C
  UUFIP(IY,IX)=RG(1)*UUR1(IY,IX)*UUR2(IY,IX)*UUADM(IY,IX)
  1 /UUL(IZ)
  UUFIP(IY,IX)=UUFIP(IY,IX)*RHO2*UUVOL(IY,IX)
  1011 CONTINUE
C
C----- SET FIP
C
  CALL SETYX(FIP,UUFIP,NY,NX)
C
  RETURN
  102 CONTINUE
C--- for CMDOT,EQ,GRND--- Inter-phase mass transfer (MDT).
C----- GET VARIABLE NEEDED: V1,V2,W1,W2,R1,R2,VOL
C
  CALL GETYX(W1,UW1,NY,NX)
  CALL GETYX(W2,UW2,NY,NX)
  CALL GETYX(R1,UUR1,NY,NX)
  CALL GETYX(R2,UUR2,NY,NX)
  CALL GETYX(VOL,UVOL,NY,NX)
C
  DO 1021 IY=1,NY
  DO 1021 IX=1,NY
C
C----- CALCULATION STARTS
C
  UUADM(IY,IX)=ABS(UW1(IY,IX)-UW2(IY,IX))
C
C--- CALCULATE MDT
C
  UUMDT(IY,IX)=RG(2)*UUR1(IY,IX)*UUR2(IY,IX)*UUADM(IY,IX)
  1 /UUL(IZ)
  UUMDT(IY,IX)=UUMDT(IY,IX)*RHO2*UUVOL(IY,IX)
  1021 CONTINUE
C
```

```

C--- SET MDT
C
    CALL SETYX(MDT,UUMDT,NY,NX)
C
    RETURN
C*****
C
C--- GROUP 19. Special calls to GROUND from EARTH
C
    19 GO TO (191,192,193,194,195,196,197,198,199,1910,1911,1912,
    1913,1914,1915,1916,1917),ISC
    191 CONTINUE
C * ----- SECTION 1 ----- START OF TIME STEP.
    IF(ISTEP.NE.1) GO TO 1911
    CALL GETZ(ZWNZ,UUZWNZ,NZ)
    DO 1911 IZ=1,NZ
        UUL(IZ)=0.01584*UUZWNZ(IZ)
    1911 CONTINUE
    RETURN
    196 CONTINUE
C * ----- SECTION 6 ----- FINISH OF IZ SLAB.
    IF(ISWEEP.NE.LSWEEP) RETURN
    CALL GETYX(W1,UW1,NY,NX)
    CALL GETYX(W2,UW2,NY,NX)
    CALL GETYX(R1,UUR1,NY,NX)
    CALL GETYX(R2,UUR2,NY,NX)
    DO 1961 IY=1,NY
        DO 1961 IX=1,NX
            UW(1Y,1X)=UW1(1Y,1X)*UUR1(1Y,1X)+UW2(1Y,1X)*UUR2(1Y,1X)
    1961 CONTINUE
    CALL SETYX(21,UW,NY,NX)
    END
$BEND

```

APPENDIX B LISTING OF PROGRAM "SPLINE"

```
PROGRAMME MAIN
C*****
C
C   THIS PROGRAM IS TO FIND A FUNCTION WHICH CAN
C   REPRESENT THE RANDOM DATA BY A BEST FITTING
C   AND SMOOTH LINE
C
C*****
C   DIMENSION R(32),R1(32),R2(32),T(32),T1(32),U(32),
1       V(32),A(32),B(32),C(32),D(32),X(32),
2       P(32),DP(32)
C   DATA NSAMP/29/
C
C*** READ DATA FROM THE DATA FILE
C
C   READ (12,101) X,Y
101  FORMAT(5(1X,1PE11.3))
C
C*** NO. OF DATA POINTS
C
C   N=NSAMP
C   N2=N+1
C
C*** (N-(2*N)**0.5).LT.S.LT.(N+(2*N)**0.5)
C
C   S=FLOAT(NSAMP)
C
C*** ESTIMATE DIVIATION
C
C   DO 10 I=1,N
10  DY(I)=0.01*Y(I)
C
C
C*****
C
C*** BEGINNING OF SPLINE FITTING
C
C   X(1)=0.
C   N1=2
C   M1=N1-1
C   M2=N2+1
C   R(M1)=0.
C   R(N1)=0.
C   R1(N2)=0.
C   R2(N2)=0.
C   R2(M2)=0.
C   U(M1)=0.
C   U(N1)=0.
C   U(N2)=0.
C   U(M2)=0.
C   Q=0.
C   M1=N1+1
C   M2=N2-1
```

```
H=X(M1)-X(N1)
F=(P(M1)-P(N1))/H
DO 100 I=M1,M2
G=H
H=X(I+1)-X(I)
E=F
F=(P(I+1)-P(I))/H
A(I)=F-E
T(I)=2.*(G+H)/3.
T1(I)=H/3.
R2(I)=DP(I-1)/H
R(I)=DP(I+1)/H
100 R1(I)=-DP(I)/G-DP(I)/H
DO 110 I=M1,M2
B(I)=R(I)*R(I)+R1(I)*R1(I)+R2(I)*R2(I)
C(I)=R(I)*R1(I+1)+R1(I)*R2(I+1)
110 D(I)=R(I)*R2(I+2)
F2=-S

C
C*****
C
C*** NEXT ITERATION
C
115 DO 120 I=M1,M2
R1(I-1)=F*R(I-1)
R2(I-2)=G*R(I-2)
R(I)=1./(Q*B(I)+T(I)-F*R1(I-1)-G*R2(I-2))
U(I)=A(I)-R1(I-1)*U(I-1)-R2(I-2)*U(I-2)
F=Q*C(I)+T1(I)-H*R1(I-1)
G=H
120 H=D(I)*Q
I=M2
130 U(I)=R(I)*U(I)-R1(I)*U(I+1)-R2(I)*U(I+2)
I=I-1
IF(I.GE.M1) GO TO 130
E=0.
H=0.
DO 140 I=N1,M2
G=H
H=(U(I+1)-U(I))/(X(I+1)-X(I))
V(I)=(H-G)*DP(I)*DP(I)
140 E=E+V(I)*(H-G)
G=-H*DP(N2)*DP(N2)
V(N2)=G
E=E-G*H
G=F2
F2=E*Q*Q
IF(F2.GE.S.OR.F2.LE.G) GO TO 160
F=0.
H=(V(M1)-V(N1))/(X(M1)-X(N1))
DO 150 I=M1,M2
G=H
H=(V(I+1)-V(I))/(X(I+1)-X(I))
G=H-G-R1(I-1)*R(I-1)-R2(I-2)*R(I-2)
F=F+G*R(I)*G
150 R(I)=G
H=E-Q*F
IF(H.LE.0.) GO TO 160
Q=Q+(S-F2)/((SQRT(S/E)+Q)*H)
GO TO 115
```

```

C
C*****
C
C*** RESULTING COEFF. AND DIFFERENTIAL
C
  160 DO 170 I=N1,N2
      A(I)=P(I)-Q*V(I)
  170 C(I)=U(I)
      DO 180 I=N1,M2
          H=X(I+1)-X(I)
          D(I)=(C(I+1)-C(I))/(3.*H)
          B(I)=(A(I+1)-A(I))/H-(H*D(I)+C(I))*H
C
C*** THE EQUATION FOR Y IS
C   Y=A(I)+B(I)*H+C(I)*H*H+D(I)*H*H*H
C   WHERE H=X-X(I) AND X(I).LE.X.LT.X(I+1)
C
C*** WRITE THE COEFFECIENTS
C
      WRITE (6,201) A
201  FORMAT(/1H ,2X,2HA=//
1      5(1X,1PE11.3))
      WRITE (6,202) B
202  FORMAT(/1H ,2X,2HB=//
1      5(1X,1PE11.3))
      WRITE (6,203) C
203  FORMAT(/1H ,2X,2HC=//
1      5(1X,1PE11.3))
      WRITE (6,204) D
204  FORMAT(/1H ,2X,2HD=//
1      5(1X,1PE11.3))
      WRITE (6,205) X
205  FORMAT(/1H ,2X,2HX=//
1      5(1X,1PE11.3))
      WRITE (6,206) Y
206  FORMAT(/1H ,2X,2HY=//
1      5(1X,1PE11.3))
      STOP
      END

```

APPENDIX C EXPERIMENTAL DATA OF NUMBER AND MEAN DIAMETER OF
DROPLETS IN CROSS-SECTIONS OF KEROSENE-INTO-WATER
STEADY JET

In tables c.1-c.9,

Z is the axial distance from the nozzle exit in mm,

d is the diameter of the size-groups in mm,

$\bar{d}_Z$ is the mean diameter of droplets in a cross-section
in mm, and

$$\text{magnification} = \frac{\text{diameter measured on screen}}{\text{real diameter}} .$$

The magnification of a negative can be obtained by
measuring the image of the ruler on the screen.

Z	d							$\bar{d}_Z$
	0-1	-2	-3	-4	-5	-6	-7	
70-90	1	1	13	27	37	21	3	5.46
-110	3	3	11	27	33	19	5	4.85
-130	2	2	20	40	41	31	4	4.72
-150	4	5	18	34	50	25	6	4.69
-170	3	4	11	26	36	23	5	4.97
-190	5	2	13	33	43	26	9	4.87
-210	-	4	13	37	49	21	5	4.69
-230	3	4	13	23	34	27	4	4.78

No of negatives: 30

Magnification: 1.27

Table c.1 The number of droplets: case 6-9

Z	d						$\bar{d}_Z$
	0-1	-2	-3	-4	-5	-6	
40-60	358	284	168	60	19	1	1.93
-80	487	366	263	121	38	-	2.78
-100	665	458	226	109	45	1	2.88
-120	828	482	265	135	49	3	2.94
-140	898	557	262	139	50	-	2.87
-160	1037	494	251	123	41	1	2.80
-180	1134	535	286	127	47	1	2.21
-200	1153	561	262	96	35	1	2.67
-220	1177	532	282	127	30	1	2.74

No of negatives: 30

Magnification: 1.27

Table c.2 The number of droplets: case 6-10

Z	d						$\bar{d}_Z$
	0-1	-2	-3	-4	-5	-6	
10-30	283	38	9	-	-	-	1.43
-50	580	201	64	13	-	-	1.85
-70	823	337	157	42	8	-	2.37
-90	1191	415	216	58	13	1	2.44
-110	1503	540	232	80	21	1	2.48
-130	1908	635	243	95	17	2	2.43
-150	2176	631	269	105	19	-	2.42
-170	2370	657	283	90	29	-	2.42
-190	2600	695	278	88	19	1	2.33
-210	2874	696	260	88	16	-	2.26

No of negatives: 24

Magnification: 1.27

Table c.3 The number of droplets: case 6-11

Z	d								$\bar{d}_z$
	0-1	-2	-3	-4	-5	-6	-7	-8	
80-100	-	2	4	19	37	26	22	3	5.49
-120	6	4	4	15	39	25	20	3	6.19
-140	3	6	9	18	26	24	24	8	5.58
-160	7	7	2	13	23	25	17	7	5.74
-180	9	10	4	28	29	25	24	9	5.68
-200	5	4	3	22	32	42	20	10	5.69
-220	5	4	8	18	29	19	17	6	5.57
-240	7	6	4	18	23	24	20	6	5.68

No of negatives: 30

Magnification: 1.17

Table c.4 The number of droplets: case 6-12

Z	d							$\bar{d}_z$
	0-1	-2	-3	-4	-5	-6	-7	
80-100	306	241	198	88	66	20	-	3.50
-120	379	288	200	84	50	15	-	3.29
-140	387	239	195	87	54	20	1	3.46
-160	375	243	215	121	64	29	-	3.56
-180	456	274	197	67	38	23	2	3.40
-200	495	243	196	95	41	25	1	3.45
-220	559	294	213	72	34	22	2	3.32
-240	562	300	217	69	43	18	2	3.29

No of negatives: 30

Magnification: 1.17

Table c.5 The number of droplets: case 6-13

Z	d						$\bar{d}_Z$
	0-1	-2	-3	-4	-5	-6	
80-100	660	542	313	94	37	9	2.85
-120	783	628	316	105	42	10	2.84
-140	1065	635	340	105	31	12	2.77
-160	1260	659	338	98	40	11	2.77
-180	1550	711	346	122	47	8	2.76
-200	1705	788	354	94	33	8	2.59
-220	1623	761	332	104	25	13	2.64
-240	1972	741	385	100	46	6	2.64

No of negatives: 30

Magnification: 1.17

Table c.6 The number of droplets: case 6-14

Z	d							$\bar{d}_z$
	0-1	-2	-3	-4	-5	-6	-7	
60-80	226	199	199	114	25	7	-	3.15
-100	240	233	204	104	39	4	-	3.14
-120	329	252	198	98	37	6	-	3.13
-140	399	237	197	85	37	7	-	3.13
-160	398	258	221	77	40	2	1	3.04
-180	429	260	190	69	32	4	-	3.13
-200	370	242	224	69	34	6	-	3.04
-220	468	257	195	75	29	2	-	2.92

No of negatives: 24

Magnification: 0.64

Table c.7 The number of droplets: case 6-15

Z	d							$\bar{d}_Z$
	0-1	-2	-3	-4	-5	-6	-7	
60-80	765	371	175	68	23	5	1	2.82
-100	1052	387	171	75	36	9	2	3.00
-120	1246	384	162	63	27	9	-	2.81
-140	1686	408	175	65	24	15	-	2.82
-160	1866	453	170	78	29	17	1	2.91
-180	2050	474	160	69	35	16	1	2.88
-200	2240	474	164	74	40	14	1	2.93
-220	2527	517	156	46	25	9	-	2.49

No of negatives: 21

Magnification: 0.64

Table c.8 The number of droplets: case 6-16

Z	d						$\bar{d}_Z$
	0-1	-2	-3	-4	-5	-6	
60-80	1228	438	170	77	30	1	2.64
-100	1602	518	183	83	20	-	2.55
-120	2119	652	184	103	32	4	2.49
-140	2725	633	185	82	33	-	2.40
-160	2911	722	187	77	33	1	2.36
-180	3187	483	176	66	29	1	2.36
-200	3648	653	172	74	26	1	2.25
-220	3847	726	163	66	26	3	2.22

No of negatives: 28

Magnification: 0.64

Table c.9 The number of droplets: case 6-17

APPENDIX D THE PHOENICS LISTINGS FOR THE KEROSENE-INTO-
WATER TRANSIENT JET

D.1 "Q1"

```
*****
*                                     *
*      KEROSENE-INTO-WATER          *
*      TRANSIENT JET                *
*                                     *
*      XI SHI-TONG                   *
*      IMPERIAL COLLEGE/CFDU         *
*      1985                          *
*                                     *
*****
```

```
GROUP 1. RUN TITLE & PHUNNI ACCOUNT NUMBER
TEXT(KEROSENE-INTO-WATER TRANSIENT JET
GROUP 2. TRANSIENCE, TIME-STEP SPECIFICATION
STEADY=.F.
GRDPWR(T,50,1.,1.)
GROUP 3. X-DIRECTION GRID SPECIFICATION
CARTES=.F.
GROUP 4. Y-DIRECTION GRID SPECIFICATION
YVLAST=4.E-02
NY=25
YFRAC(1)=-3.
YFRAC(2)=0.00666
YFRAC(3)=3.
YFRAC(4)=0.01
YFRAC(5)=9.
YFRAC(6)=0.02
YFRAC(7)=6.
YFRAC(8)=0.03
YFRAC(9)=4.
YFRAC(10)=0.147
GROUP 5. Z-DIRECTION GRID SPECIFICATION
GRDPWR(Z,32,2.E-01,1.5)
GROUP 7. VARIABLES STORED,SOLVED & NAMED
ONEPHS=.FALSE.
SOLUTN(P1,Y,Y,N,N,N,Y)
SOLUTN(V1,Y,Y,N,N,N,Y)
SOLUTN(V2,Y,Y,N,N,N,Y)
SOLUTN(W1,Y,Y,N,N,N,Y)
SOLUTN(W2,Y,Y,N,N,N,Y)
SOLUTN(R1,Y,Y,N,N,N,Y)
SOLUTN(R2,Y,Y,N,N,N,Y)
NAME(21)=W
SOLUTN(21,Y,N,N,N,N,N)
GROUP 9. PROPERTIES OF THE MEDIUM (OR MEDIA)
RH01=785.
RH02=998.
*** EFFECTIVE VISCOSITY MODEL ENUL=0.02845*W0*R0
ENUT=4.279E-5
GROUP 10. INTER-PHASE-TRANSFER PROCESSES AND PROPERTIES
```

```
CFIPS=GRND
  GROUP 11. INITIALIZATION OF VARIABLE OR POROSITY FIELDS
FIINIT(R1)=1.E-10
FIINIT(R2)=1.
  GROUP 13. BOUNDARY CONDITIONS AND SPECIAL SOURCES
    JET EXIT
    PATCH(1,LOW,1,1,1,3,1,1,1,1000)
    COVAL(1,P1,FIXFLU,GRND)
    COVAL(1,W1,ONLYMS,GRND)
    FREE STREAM AT THE JET EXIT PLANE
    PATCH(2,LOW,1,1,4,25,1,1,1,1000)
    COVAL(2,P2,FXP,0.0)
    COVAL(2,W2,ONLYMS,0.0)
    COVAL(2,W1,ONLYMS,0.0)
    FREE STREAM
    PATCH(3,NORTH,1,1,25,25,1,32,1,1000)
    COVAL(3,P2,FXP,0.0)
    COVAL(3,W2,ONLYMS,0.0)
    COVAL(3,W1,ONLYMS,0.0)
    OUTLET OF JET
    PATCH(4,CELL,1,1,1,25,32,32,1,1000)
    COVAL(4,P1,FXP,0.0)
    COVAL(4,P2,FXP,0.0)
    BOUANCY
    PATCH(5,PHASEM,1,1,1,25,1,32,1,1000)
    COVAL(5,W1,FIXFLU,2.66)
    GROUP 14. DOWNSTREAM PRESSURE FOR PARAB=.TRUE.
    GROUP 15. TERMINATION OF SWEEPS
    LSWEET=10
    RESREF(P1)=1.E-8
    RESREF(V1)=1.E-8
    RESREF(V2)=1.E-8
    RESREF(W1)=1.E-8
    RESREF(W2)=1.E-8
    RESREF(R1)=1.E-8
    RESREF(R2)=1.E-8
    GROUP 16. TERMINATION OF ITERATIONS
    GROUP 17. UNDER-RELAXATION DEVICES
    RELAX(V1,FALSDT,0.1)
    RELAX(V2,FALSDT,0.1)
    RELAX(W1,FALSDT,0.001)
    RELAX(W2,FALSDT,0.001)
    GROUP 19. EARTH CALLS TO GROUND SUBROUTINE
    *** RG(1) IS CF
    RG(1)=0.1
    *** RG(2) IS THE DROPLET SIZE
    RG(2)=2.14E-3
    *** RG(3) IS THE DIA. OF THE NOZZLE
    RG(3)=1.6E-3
    GROUP 22. MONITOR PRINT-OUT
    NPRMON=10
    NPRMNT=10
    ITABL=0
    IPLTL=25
    NUMCLS=10
    NCOLCO=10
    GROUP 23. FIELD PRINT-OUT & PLOT CONTROL
    NTPRIN=20
    OUTPUT(P1,Y,Y,Y,Y,Y,Y)
    OUTPUT(W1,Y,Y,Y,Y,Y,Y)
```

OUTPUT(W2,Y,Y,Y,Y,Y,Y)
OUTPUT(R1,Y,Y,Y,Y,Y,Y)
OUTPUT(Z1,Y,Y,Y,Y,Y,Y)
STOP

D.2 "GROUND"

```

$BATCH
C$DIR**MAIN
C FILE NAME GROUND.FTN-----170785
C
C   GROUND.FTN TO BE USED IN CONJUNCTION WITH Q FILES
C   WHICH SOLVES THE PROBLEM OF KEROSENE-INTO-WATER
C   TRANSIENT JET
C
C   XI SHI-TONG
C   IMPERIAL COLLEGE/CFDU/1985
C
C*****
      SUBROUTINE GROUND
$INCLUDE SATEAR/G
$INCLUDE GRDLOC/G
$INCLUDE GRDEAR/G
CXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX USER SECTION STARTS:
C
C 1   Set dimensions of satellite-to-GROUND data arrays to those
C     of the satellite.
C     COMMON/LGRND/LG(10)/IGRND/IG(10)/RGRND/RG(10)/CGRND/CG(10)
C     LOGICAL LG,USEGRD
C     CHARACTER*4 CG
C
C 2   User dimensions own arrays here, for example:
C     DIMENSION UUH(10,10),UUC(10,10),UUX(10,10),UUZ(10)
C----- 2-D ARRAYS
C     DIMENSION U UW1(25,1),UW2(25,1),UUR1(25,1),UUR2(25,1)
C     DIMENSION U UW(25,1),UVOL(25,1),URV2D(25,1),UADM(25,1)
C     DIMENSION UUFIP(25,1),UUPP1(25,1),UWW1(25,1),UUA2D(25,1)
C
C--- 1-D ARRAY AT Y-DIRECTION
C
C     DIMENSION UUA1D(25)
C
C----- 1-D ARRAYS AT Z-DIRECTION
C
C     DIMENSION UUZWNZ(32),UUA1Z(32)
C
C*** 1-D ARRAYS AT T-DIRECTION
C
C     DIMENSION UUTT(200),UUL0(200),UW11(200)
C
C*** 1-D ARRAYS FOR VOLUME FLOW RATE-TIME CORRELATION AT THE EXIT
C
C     DIMENSION UUA(30),UUB(30),UUC(30),UUD(30),UUX(30)
C
C
C----- END
C
C
C 3   User places his data statements here, for example:
C     DATA NXDIM,NYDIM/10,10/
C

```

C*** DATA FOR EXIT B.C.

C

DATA UUA/0.0,0.3874,0.7151,1.033,1.333,1.612,1.869,2.103,

1 2.315,2.505,2.674,2.823,2.953,3.065,3.163,3.247,

2 3.321,3.385,3.442,3.492,3.537,3.576,3.612,3.643,

3 3.671,3.696,3.717,3.737,3.754,3.771/

DATA UUB/0.0,3.296,3.241,3.096,2.899,2.682,2.456,2.231,

1 2.010,1.795,1.587,1.390,1.209,1.048,0.9074,0.7875,

2 0.6862,0.6022,0.5315,0.4719,0.4205,0.3748,0.3341,

3 0.2968,0.2621,0.2309,0.2043,0.1834,0.1695,0.0/

DATA UUC/0.0,0.0,-0.5488,-0.9057,-1.057,-1.120,-1.136,

1 -1.117,-1.089,-1.062,-1.017,-0.9549,-0.8563,-0.7539,

2 -0.6493,-0.5515,-0.4592,-0.3823,-0.3232,-0.2730,

3 -0.2419,-0.2142,-0.1933,-0.1798,-0.1670,-0.1453,

4 -0.1202,-0.08929,-0.04909,0.0/

DATA UUD/0.0,-1.829,-1.190,-0.5058,-0.2089,-0.05285,0.06428,

1 0.09343,0.08921,0.1487,0.2078,0.3288,0.3414,0.3486,

2 0.3258,0.3077,0.2564,0.1972,0.1672,0.1037,0.09234,

3 0.06980,0.04495,0.04262,0.07222,0.08379,0.1030,0.1340,

4 0.1636,0.0/

DATA UUNXX/30/

DATA UUX/0.0,0.1,0.2,0.3,0.4,0.5,0.6,0.7,0.8,0.9,1.0,1.1,

1 1.2,1.3,1.4,1.5,1.6,1.7,1.8,1.9,2.0,2.1,2.2,2.3,2.4,

2 2.5,2.6,2.7,2.8,2.9/

C

IF(IGR.EQ.13) GO TO 13

GO TO (1,2,3,4,5,6,7,8,9,10,11,12,13,14,15,16,17,18,19,20,21,
122,23,24),IGR

C*****

C

C--- GROUP 1. Run title

C

1 CONTINUE

CALL MAKE(VOL)

CALL MAKE(RV2D)

RETURN

C*****

C

C--- GROUP 10. Inter-phase-transfer processes and properties

C

10 GO TO (101,102,103,104),ISC

101 CONTINUE

C--- for CFIPS.LE.GRND--- Inter-phase friction coefficient (FIP).

C----- GET VARIABLE NEEDED: W1,W2,R1,R2,VOL

C

CALL GETYX(W1,UW1,NY,NX)

CALL GETYX(W2,UW2,NY,NX)

CALL GETYX(R1,UUR1,NY,NX)

CALL GETYX(R2,UUR2,NY,NX)

CALL GETYX(VOL,UUVOL,NY,NX)

C

C--- CALCULATION STARTS

C

DO 1011 IY=1,NY

DO 1011 IX=1,NX

UADM(IY,IX)=ABS(UW2(IY,IX)-UW1(IY,IX))

C

C----- CALCULATE FIP TO BE PUT IN THE CODE

C

UUCFI=RG(1)

```

      IF(IZSTEP.EQ.1) UUCF1=500
      UUFIP(IY,IX)=UUCF1*UUR1(IY,IX)*UUR2(IY,IX)*UUADM(IY,IX)/
1      RG(2)
      UUFIP(IY,IX)=UUFIP(IY,IX)*RH02*UUVOL(IY,IX)
1011 CONTINUE
C
C----- SET FD
C
      CALL SETYX(FIP,UUFIP,NY,NX)
C
C----- END OF INTER-FLUID FRICTION CORRELATION
C
      RETURN
C*****
C
C--- GROUP 13. Boundary conditions and special sources
C
      13 CONTINUE
      GO TO (130,131,132,133,134,135,136,137,138,139,1310,
1311,1312,1313,1314,1315,1316,1317,1318,1319,1320,1321),ISC
1311 CONTINUE
C----- value = GRND
C
C*** CALL FOR TIME-DEPENDENT BOUNDARY CONDITIONS
C
      IF(IZSTEP.GT.1) RETURN
C
C*** CALCULATE THE VOLUME FLOW RATE AT THE EXIT
C
      DO 3301 I=1,UUNXX
      IF(UUX(I).LE.UUTT(ISTEP).AND.UUTT(ISTEP).LT.UUX(I+1))
1      GO TO 3302
3301 CONTINUE
3302 CONTINUE
      UJML0(ISTEP)=UUA(1)+UUB(1)*(UUTT(ISTEP)-UUX(1))+UUC(1)*
1      (UUTT(ISTEP)-UUX(1))**2.+UUD(1)*
2      (UUTT(ISTEP)-UUX(1))**3.
C
C*** SPECIAL FOR TIME < UUX(2)
C
      UJML1=UUA(2)
      IF(I.NE.1) GO TO 3303
      UJML0(ISTEP)=UJML1*UUTT(ISTEP)/UUX(2)
3303 CONTINUE
C
C*** UJARE IS THE AREA OF THE JET EXIT
C
      UJARE=0.25*3.1416*RG(6)**2.
      UJML0(ISTEP)=UJML0(ISTEP)*1.E-6*RH01/UJARE
      UJW1(ISTEP)=UJML0(ISTEP)/RH01
C
C*** SET TO B.C.
C
C
      CALL ONLYIF(P1,P1,1,1,DO)
      DO 2301 IY=1,3
      DO 2301 IX=1,1
      UJPP1(IY,IX)=UJML0(ISTEP)
2301 CONTINUE
      CALL SETYX(VAL,UJPP1,3,1)

```

```
C
  CALL ONLYIF(W1,W1,1,1,DO)
  DO 2302 IY=1,3
  DO 2302 IX=1,1
  UWW1(IY,IX)=UWL0(ISTEP)/RHO1
2302 CONTINUE
  CALL SETYX(VAL,UWW1,3,1)
C
  RETURN
C*****
C
C--- GROUP 19. Special calls to GROUND from EARTH
C
  19 GO TO (191,192,193,194,195,196,197,198,199),ISC
  191 CONTINUE
C * ----- SECTION 1 ----- START OF TIME STEP.
  IF(ISTEP.NE.1) GO TO 1911
  CALL GETZ(ZWNZ,UZWZ,NZ)
  DO 1912 IZ=1,NZ
  UUL(IZ)=0.01584*UZWZ(IZ)
  1912 CONTINUE
C
  1911 CONTINUE
  IF(ISTEP.NE.1) GO TO 2911
  UUTL=0.
  GO TO 2912
  2911 UUTL=UUTL+DT
  2912 CONTINUE
  UUT(ISTEP)=UUTL
  2914 CONTINUE
  RETURN
  196 CONTINUE
C * ----- SECTION 6 ----- FINISH OF IZ SLAB.
  IF(ISWEEP.NE.LSWEEP) RETURN
  CALL GETYX(W1,UW1,NY,NX)
  CALL GETYX(W2,UW2,NY,NX)
  CALL GETYX(R1,UUR1,NY,NX)
  CALL GETYX(R2,UUR2,NY,NX)
  DO 1961 IY=1,NY
  DO 1961 IX=1,NX
  UWW(IY,IX)=UW1(IY,IX)*UUR1(IY,IX)+UW2(IY,IX)*UUR2(IY,IX)
  1961 CONTINUE
  CALL SETYX(21,UUV,NY,NX)
C
C*** CALCULATE THE MASS FLOW RATE OF PHASE-1
C
  UUMA1D(1)=3.1416*RHO1*UUR1(1,1)*UW1(1,1)*UURV2D(1,1)**2.
  DO 1961 IY=2,NY
  1961 UUMA1D(IY)=3.1416*RHO1*UUR1(IY,1)*UW1(IY,1)*(UURV2D(IY,1)**
1      2.-UURV2D(IY-1,1)**2.)
  UUMA2D(1,1)=UUMA1D(1)
  DO 1962 IY=2,NY
  1962 UUMA2D(IY,1)=(UUMA2D(IY-1,1)+UUMA1D(IY))
C
C*** CALCULATE      FOR EACH IZ-STATION
C
  UUMA1Z(IZSTEP)=UUMA2D(NY,1)
  WRITE(6,*) "ISTEP=",ISTEP
  WRITE(6,*) "TLAST=",UUTL
  WRITE(6,1987) UUMA1Z
```

```
1987 FORMAT(/1H .2X.5HMAIZ=//  
1      5(1X.1PE11.3))  
      END  
$BEND
```