

COMPUTER AND WATER-MODEL SIMULATION OF FLOW THROUGH
POPPET VALVES

PHD THESIS

H Q QIN

11 January 1985

E R R A T A L I S T

Page 3	Line 4	for 'phenomena' read 'phenomenon'.
Page 24	Fig 1.1-3	for ' θ ' read ' ϕ '.
Page 26	Line 17	delete sentence after 'at large lifts', insert 'because of its inertia, fluid flowing along a surface cannot be made to follow a sharp edge, but breaks away from the surface to form a free jet. The discharge coefficient was decreased".
Page 41	Line 3	insert 'been' after 'has'.
Page 43	Line 4	for 'As' read 'At'.
Page 44	Eqn 2.5-3 Eqn 2.5-4 Eqn 2.5-5	U_T should be in the lower case, u_T . read ' k^2 ' for ' κ^2 '. read ' κ ' for ' k '.
Page 45	Line 7 Eqn 2.5-8	read lower case u_w for U_w ; the error appears twice in the same line on this page. read κ for k .
Page 55	Eqn 3.4-1 Eqn 3.4-2 Lines 20 & 21	should be numbered 3.23-1 should be deleted. the words "It is computed from:" should be deleted.
Page 56	Lines 1,2 & 3	should be deleted.
Page 58	Eqn 3.4-3	should be numbered 3.4-1.
Page 59	Eqn 3.4-4	should be numbered 3.4-2.

	Eqn 3.4-5	should be numbered 3.4-3.			
Page 60	Eqn 3.4-6 Eqn 3.4-7	should be numbered 3.4-4. should be numbered 3.4-5.			
Page 65	Line 10 Line 11 Line 13 Line 17	insert $(\leq 0.003 \cdot U_{in}^2)$ after $1.2 \text{ m}^2/\text{s}^2$. read m^2/s^3 for m^2/s^2 , and add $(\leq 0.09 \cdot k_{in}^{3/2} / (0.03 \cdot D/2))$. read N/m^2 for kg/m^2 . (This error also appears on Page 90, Line 12, and Page 113, Line 9.) read u for w .			
Page 69	Last line	read '4.5-2' for '4.5-1'.			
Page 70	Table 4.5-2	replace 'Inertial under-relaxation by 'DTFALS'.			
Page 72	Flg 4.6-1	the word 'aurface' should read 'surface'.			
Page 76	Line 5	delete 'with the difference, p_d , between the local pressure and the one at the pipe junction' and insert 'In the form of the pressure difference, p_d , which is defined by the local pressure minus the one at the pipe junction point'.			
Page 87	Lines 17 & 18	the words 'former' and 'latter' should be interchanged.			
Page 89	Table 5.3-1	Insert another line at the bottom: <table border="1"> <tr> <td>L</td> <td>0.1</td> <td>0.1</td> </tr> </table>	L	0.1	0.1
L	0.1	0.1			
Page 91	Lines 17 & 18	delete 'that 20x30 grid-nodes' and insert 'that 8 grid-nodes in region A (20x30 for the total grid-nodes)'.			
Page 96	Last line	delete ' $L_v D_v$ ' and insert ' L_v/D_v '.			
Page 97	Line 15	delete 'inlet' and insert 'outlet'.			
Page 98	Lines 2 & 5	delete ' $L_v D_v$ ' and insert ' L_v/D_v ' in all cases.			
Page 105	Line 16	read 'fills' for 'fill'.			

Page 111	Flg 6.3-1	insert ' D_0 ' above ' $d/2$ '.
Page 136	Flg 6.6-13	delete ' Δ prediction' and insert ' Δ experimental data of Annand'.
Page 137	Line 5	delete bracket after ' L_V/D_V '.
Page 141	Line 17	read 'value of' for 'values'.
Page 142	Line 10	read 'satisfactorily in most cases' for 'satisfactorilly'.
Page 154	Table 7.3-1	'0.002' should be '0.05' in all cases.
Page 174	Line 4	read 'centripetal' for 'centrifugal'.
Page 182	Table 8.22-1	'0.00/m' should read '0.001m'.
Page 206	Line 6	'seated' and 'seatless' should be interchanged.
Pages 243	to 248	'Ean' should read 'Eqn' in all cases.
Page 253	Eqn B-4	read ' ϕ_1 ' for ' ϕ '.

**COMPUTER AND WATER-MODEL SIMULATION OF FLOW
THROUGH POPPET VALVES**

BY

H Q QIN

1984

**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.**

ABSTRACT

An existing computer code embodying a two-equation ($k-\epsilon$) turbulence-model has been used to simulate two-dimensional steady flows through axisymmetric pipe expansions, including cases with and without the partial obstruction by a poppet valve. Steady and unsteady laminar flows through the valves are also considered.

In addition, the effect of valve geometry on the discharge coefficient is investigated. Satisfactory agreement is obtained between the predicted values of the discharge coefficient and available experimental data. The computed flow-fields all appear to be qualitatively realistic.

A water-model rig of a valve port and cylinder has been constructed. This has allowed details of the flow pattern near the plane of enlargement to be made visible. The computer simulation of such flows has been compared with photographs of vortices taken from the experimental rig. The predictions appear satisfactorily in agreement with the measurements.

The experiments have also revealed some important features. In particular, it has been observed that the mixing process in the circumferential direction, especially within the cores of toroidal vortices takes place with unexpected rapidity. This "diffusion" cannot be reliably accounted for with the limited information available. However, a plausible theory has been advanced to explain the phenomenon.

PREFACE

I joined the Computational Fluid Dynamics Unit (CFDU, which until 1982 was the Heat Transfer Section, Department of Mechanical Engineering) of Imperial College, as a research student, in 1980. The work reported in this thesis is mainly concerned with fluid flows through poppet valves computationally and experimentally.

During the first six months of my studies, I followed the M.Sc. courses given by the Heat Transfer Section and worked on the numerical modelling of film cooling using the GENMIX computer code. The experience obtained during this period proved to be valuable for my later research work.

In the next stage, I learned to use and modify the CORA 2 computer code of Professor Spalding. Some of the work included calculation of turbulent flows in internal combustion engines. It involved both hydrodynamic and combustion processes.

During this period under Professor Spalding's guidance, the importance of the intake valve flow became prominent; the work started with the use of PHOENICS computer code for the prediction of a well-documented problem: axisymmetric expansion. This served also to examine the validity of the turbulence model, the wall functions in PHOENICS code and the accuracy of the porosity method. Then followed, a series of studies on turbulent air flows through various poppet valves, seatless as well as seated ones. Some of these works were reported as CFDU PDR (PHOENICS Demonstration Report).

The prime interest was in the investigation of the fundamentals of valve flow, and to this end a water model rig was constructed for studying the important details of the laminar flow-pattern around the valve. This led to the discovery of an unexpected phenomena which was subsequently reported and published.

I would now like to acknowledge my debt of gratitude to all those who have provided valuable assistance during my years at Imperial College. First and foremost, I am deeply grateful to my supervisors, Professor D B Spalding and Dr A S C Ma, for their skillful guidance and constant encouragement. They have guided me not only in the main direction of my research topic, but also helped me on how to overcome difficulties arising from various aspects of the work, including report writing. It is indeed a privilege to have worked with them.

I should also like to thank Dr W M Pun for his very useful discussions, particularly on the basic prediction procedures used in the CFDU unit and in technical writing methodology. His suggestions and willingness to help is highly appreciated. I am very grateful.

At the beginning of my research, friendly discussions with Drs Andrzej Przekwas, L. T. Tam, A. Abdelmeguid and R. L. T. Sun were invaluable.

Assistance in setting up the experimental rig given by Jeremy Wu deserves a special mention here. He shared his time and experience unselfishly with me, even at weekends and during late evenings, for which I am greatly indebted.

I also enjoyed the friendship of, and discussions with, Simon Lo. Informal discussions with other colleagues - Dr J. O. Ilegbusi, W. Fan, P. Shepherd, M. Andrews and S. Parameswaran etc. were too important to escape mention.

Thanks are due to Mrs Frith Oliver for her assistance in various administrative matters. The excellent typing of the thesis by Mrs Oliver and Mrs Catherine Webb, and their patience and skill in checking every detail are greatly appreciated.

The help of Mr Bob King on all aspects of my experimental programme and his endeavour in the laboratory were highly appreciated. So too am I grateful to Mr Peter Dale for helping me with computer problems.

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

Space does not permit me to list all the names of individuals and organizations who contributed towards my work. For over a period during which I have carried out my research I have met many people who have given me a helping hand in some small way. It is to all these people that I wish to offer my sincere thanks.

I also wish to acknowledge the Educational Ministry of China and the Peking Institute of Aeronautics and Astronautics for making it all possible.

<u>CONTENTS</u>	<u>PAGE</u>
	<u>NUMBER</u>
<u>ABSTRACT</u>	1
<u>PREFACE</u>	2
<u>CONTENTS</u>	5
<u>LIST OF FIGURES</u>	11
<u>LIST OF TABLES</u>	19
1. <u>INTRODUCTION</u>	21
1.1 The Problem	21
1.2 Practical Relevance	23
1.3 Previous Work	25
1.31 Experiments	25
1.32 Theoretical Calculations	27
1.4 Objectives of the Present Study	28
1.5 Present Contributions	29
1.6 Outline of the Thesis	30
2. <u>THE MATHEMATICAL PROBLEM</u>	32
2.1 Introduction	32
2.2 Partial-Differential Equations	33
2.3 "k- ϵ " Turbulence Model	37
2.4 Auxilliary Relations	40
2.5 Boundary and Initial Conditions	42
2.6 Closure	46
3. <u>THE NUMERICAL SOLUTION PROCEDURE</u>	47
3.1 Introduction	47
3.2 Formulation of the Finite-Domain Equations	48

	<u>PAGE</u>
	<u>NUMBER</u>
3.21 The Finite-Domain Grid	48
3.22 Derivation of the Finite-Domain Equations	51
3.23 Incorporation of Boundary Conditions Into the Finite-Domain Equation	55
3.3 The Solution of the Finite-Domain Equation	56
3.31 General Features of the Solution Method	56
3.32 Sequence of the Operations in the Solution Procedure	57
3.4 Control and Monitoring of the Solution Procedure	58
3.5 Closure	60
4. <u>FLOW OF AIR THROUGH AN AXISYMMETRIC ENLARGEMENT</u>	61
4.1 Introduction	61
4.2 The Problem to be Predicted	62
4.3 Details of Geometry and Dimensions	62
4.4 Boundary Conditions	65
4.5 Computational Details	66
4.51 Grid Selection	66
4.6 Presentation of Results	71
4.61 Abrupt Expansion	71
4.62 Expansion with a half angle of 45°	78
4.7 Discussion	82

	<u>PAGE</u>
	<u>NUMBER</u>
4.8 Closure	85
5. <u>AIR FLOW THROUGH A SEATLESS POPPET VALVE</u>	85
5.1 Introduction	85
5.2 Phenomena to be Predicted	87
5.3 Geometry and Dimensions	87
5.4 Boundary Conditions	90
5.5 Computational Details	91
5.51 Grid-Dependence Study	91
5.52 Other Computational Details	94
5.6 Presentation of the Main Results	95
5.7 Discussion	104
5.8 Closure	106
6. <u>FLOW OF AIR THROUGH A VALVE WITH AN</u>	
<u>ANGLED SEAT</u>	107
6.1 Introduction	107
6.2 Phenomena to be Predicted	108
6.3 Geometry, Dimensions and Cases	
Considered	110
6.4 Boundary Conditions	112
6.5 Grid Refinement	115
6.6 Presentation of the Main Results	120
6.61 Radial distribution Axial Velocity	120
6.62 Radial Velocity	125
6.63 Centreline distribution of Axial Velocity	125
6.64 Streamline Pattern and Recirculating Mass	
Flow Rate	125

	<u>PAGE</u>
	<u>NUMBER</u>
6.65 Pressure	130
6.66 Turbulent Kinetic Energy	132
6.67 Discharge Coefficient	134
6.7 Discussion	139
6.71 The Characteristics of the Flow Field	139
6.72 Turbulence Quantity	140
6.73 The Discharge Coefficient	141
6.8 Closure	142
7. <u>THE EXPERIMENTAL INVESTIGATION: FLOW OF</u>	
<u>WATER INTO A CYLINDER THROUGH A COAXIAL</u>	
<u>VALVE-PORT</u>	143
7.1 Introduction	143
7.2 Description of the Experiments	144
7.21 Apparatus	144
7.22 Procedure	149
7.3 Tests Carried Out	152
7.4 Steady Flow Tests - Observation and	
Discussion	154
7.5 Unsteady Flow Tests - Observation and	
Discussion	157
7.6 An Unexpected Phenomenon	169
7.7 Closure	177
8. <u>NUMERICAL SIMULATION OF THE EXPERIMENT</u>	178
8.1 Introduction	178
8.2 Steady-Flow Computations	180
8.21 The Phenomena to be Predicted	180

	<u>PAGE</u>
	<u>NUMBER</u>
8.22 Geometry and Dimensions	182
8.23 Flow-Domain for Computation and Grid-Distribution	182
8.24 Boundary Conditions	185
8.25 Results and Discussion	188
8.3 Unsteady-Flow Computations	197
8.31 The Phenomena to be Predicted	197
8.32 Geometry and Dimensions	197
8.33 The Flow-Domain for Calculation	199
8.34 The Use of a Composite Grid. Consisting of a 'Moving' Part and a Fixed Part	201
8.35 Boundary and Initial Conditions	203
8.36 Results and Discussion	205
8.4 Closure	219
9. <u>CONCLUSIONS</u>	220
9.1 Theoretical Predictions	220
9.2 Experiments	221
9.3 Suggestion for Future Work	222
<u>REFERENCES</u>	224
<u>NOMENCLATURE</u>	243
<u>APPENDIX A:</u> Definition of the Discharge Coefficient	250
<u>APPENDIX B:</u> Transformation of Equations for a Composite Grid	251
<u>APPENDIX C:</u> Particle Tracking	256
<u>APPENDIX D:</u> Coding of SATELLITE	258

	<u>PAGE</u>
	<u>NUMBER</u>
<u>APPENDIX E:</u> Coding of GROUND	261
<u>APPENDIX F:</u> Coding of TRACK	266

LIST OF FIGURES

<u>FIGURE</u> <u>NUMBER</u>	<u>TITLE</u>	<u>PAGE</u> <u>NUMBER</u>
1.1-1	A seatless-valve assembly	22
1.1-2	A seated-valve assembly	22
1.1-3	A typical inlet valve assembly	24
3.21-1	A two-dimensional control cell	50
3.21-2	Grid showing location of variables and their control volume	52
4.3-1	The geometrics of the axisymmetric expansions with half angles of 90° and 45°.	64
4.5-1	The velocity profile at 0.75 of $(x-l)/D_0$ for different sets of grid numbers	67
4.5-2	The influence of grid number on the velocity at $A(x-l)/D_0 = 0.75$, $Y/D_0 = 0.7$	68
4.5-3	The grid network for the abrupt expansion	68
4.6-1	Pattern of streamlines, abrupt expansion	72
4.6-2	The mean velocity at different cross sections, abrupt expansion	73
4.6-3	Spatial distribution of turbulence energy, abrupt expansion	75
4.6-4	Spatial distribution of mean pressure, abrupt expansion	77

<u>FIGURE</u> <u>NUMBER</u>	<u>TITLE</u>	<u>PAGE</u> <u>NUMBER</u>
4.6-4(A)	Radial distribution of mean pressure, at different streamwise locations	77
4.6-5	Pattern of streamlines, 45° expansion	79
4.6-6	The mean velocity at different cross- sections, 45° expansion	80
4.6-7	Spatial distribution of mean pressure, 45° expansion	81
5.1-1	Inlet seatless valve assembly	86
5.3-1	Geometrics of the seatless-valve	88
5.5-1	Grid distribution of the calculation domain	92
5.5-2	The influence of grid number on the discharge coefficient	93
5.6-1	Radial profiles of axial velocity at six stream-wise locations for Case A	96
5.6-2	Centrelle profile of axial velocity for Case A	98
5.6-3	Centrelle profiles of axial velocity for flow through a seatless valve, with three valve locations	98
5.6-4	Streamlines of flow through a seatless valve	99
5.6-5	Axial variation of wall-pressure	99
5.6-6	Centrelle values of turbulence Intensity	101

<u>FIGURE</u>	<u>TITLE</u>	<u>PAGE</u>
<u>NUMBER</u>		<u>NUMBER</u>
5.6-7	Centrelne values of turbulent kinetic energy	101
5.6-8	Comparison of Discharge Coefficient of Closed and Open Flow Situations	103
6.2-1	Mean streamline pattern for a valve flow	109
6.3-1	The geometry of an inlet valve	111
6.3-2	The details of the valve parameter	111
6.4-1	Boundary region distribution	114
6.5-1	Grid distribution	116
6.5-2	The influence of grid number of the discharge coefficient	119
6.6-1	The chosen location for the velocity sketch	121
6.6-2	Radial profiles of axial velocity at six locations for $L_v/D_v = 0.106$	122
6.6-3	Radial profiles of axial velocity at six locations for $L_v/D_v = 0.16$	123
6.6-4	Radial profiles of axial velocity at six locations for $L_v/D_v = 0.284$	124
6.6-5	Centrelne profile of axial velocity for flow through a valve with 45° angled seat with three valve locations	126
6.6-6	Radial mean velocity distribution at lift region for three lifts of 0.284, 0.16 and 0.106.	127

<u>FIGURE</u> <u>NUMBER</u>	<u>TITLE</u>	<u>PAGE</u> <u>NUMBER</u>
6.6-7	Streamlines of flow through a seated valve, $L_v/D_v = 0.16$, $W/D_v = 0.035$ $\theta = 45^\circ$	127
6.6-8(a)	Variation of the recirculation mass flow rate with axial distance for three lifts	129
6.6-8(b)	Streamline pattern in the recirculation zone	129
6.6-9(b)	The axial variation of the wall-pressure downstream of the valve for three lifts	131
6.6-9(b)	Calculated isobars for flow through a valve with an angled seat	131
6.6-10	Centreline values of turbulence intensity for three lifts	133
6.6-11	Centreline values of turbulent kinetic energy for three lifts	133
6.6-12	The effect of the lift on the discharge coefficient	135
6.6-13	The effect of the seat-width on the discharge coefficient	136
6.6-14	The effect of the seat angle on C_D	138
7.21-1(a)	Experimental rig	145
7.21-1(b)	Photograph of the experimental rig	146
7.21-2	Intake valve-port geometry used in the experiment	147
7.21-3(a)	Arrangement of mirror used to the top	

<u>FIGURE</u> <u>NUMBER</u>	<u>TITLE</u>	<u>PAGE</u> <u>NUMBER</u>
	view of the flow pattern	150
7.21-3(b)	Photograph of the mirror part of the rig	151
7.4-1	Steady flow test $Re = 400, L_V/D_V = 0.14, \theta = 0^\circ$	156
7.4-2	Steady flow test $Re = 620, L_V/D_V = 0.14, \theta = 0^\circ$	156
7.4-3	Steady flow test $Re = 1000, L_V/D_V = 0.14, \theta = 0^\circ$	156
7.4-4	Steady flow test $Re = 620, L_V/D_V = 0.14, \theta = 45^\circ$	156
7.5-1	Development of the flow pattern around a valve with seat angle of 45° $Re = 620, L_V/D_V = 0.14$	158
7.5-2	Development of the flow pattern around a valve with 45° seat angle $Re = 400, L_V/D_V = 0.14$	159
7.5-3	Development of the flow pattern around a valve with 45° seat angle $Re = 1000, L_V/D_V = 0.14$	160
7.5-4	Development of the flow pattern around a valve with 45° seat angle $Re = 620, L_V/D_V = 0.08$	164
7.5-5	Development of the flow pattern around a valve with 45° seat angle $Re = 620, L_V/D_V = 0.14$	165

<u>FIGURE</u> <u>NUMBER</u>	<u>TITLE</u>	<u>PAGE</u> <u>NUMBER</u>
7.5-6	Development of the flow pattern around a valve with 45° seat angle Re = 620, $L_v/D_v = 0.2$	166
7.5-7	Development of the flow pattern around a valve with 45° seat angle Re = 620, $L_v/D_v = 0.24$	167
7.5-8	Development of the flow pattern around a seatless valve Re = 620, $L_v/D_v = 0.14$	168
7.6-1	Top view of flow pattern showing large circumferential movement	170
7.6-2	Sketch showing the circumferential movement of the dye	171
7.6-3	Sketch showing the position of the dye injection from the top view	173
7.6-4	The inter-face line of a propagating Kelvin wave	174
7.6-5	Sketch of the explanation of the circumferential motion	175
8.21-1	Flow through a coaxial valve port geometry and dimensions	181
8.23-1	The calculation domain and a typical grid distribution	184
8.24-1	The boundaries of flow domain	187
8.25-1	Axial velocity profiles at four locations Re = 400, $L_v/D_v = 0.14$, $\theta = 0^\circ$	190

<u>FIGURE</u>	<u>TITLE</u>	<u>PAGE</u>
<u>NUMBER</u>		<u>NUMBER</u>
8.25-2	Axial velocity profiles at four locations $Re = 620, L_V/D_V = 0.14, \Theta = 0^\circ$	190
8.25-3	Axial velocity profiles at four locations $Re = 1000, L_V/D_V = 0.14, \Theta = 0^\circ$	191
8.25-4	Axial velocity profiles at four locations $Re = 620, L_V/D_V = 0.14, \Theta = 45^\circ$	191
8.25-5	The predicted velocity vectors for flow through a seatless valve with three different Reynolds Numbers of 400, 620 and 1000. $L_V/D_V = 0.14$	193
8.25-6	Predicted streamline pattern near a seatless valve. For three different Reynolds Number of 400, 620 and 1000 $L_V/D_V = 0.14$	194
8.25-7	Predicted streamline pattern near a seated valve. $Re = 620, L_V/D_V = 0.14, \Theta = 45^\circ$	195
8.32-1	Unsteady flow through a coaxial valve-port geometry and dimensions	198
8.33-1	Flow-domain for calculation	200
8.34-1	Two domains for calculation and grid distribution	202
8.35-1	The boundaries of flow domain	204

<u>FIGURE</u>	<u>TITLE</u>	<u>PAGE</u>
<u>NUMBER</u>		<u>NUMBER</u>
8.36-1	Axial-velocity profiles at different-time intervals for a seatless valve $Re = 620, L_v/D_v = 0.14, \theta = 0$	207
8.36-2	Axial-velocity profiles at different-time intervals for a valve with an angled seat $Re = 620, L_v/D_v = 0.14, \theta = 45^\circ$	208
8.36-3 (1-4)	Filament of dye particles for a seatless valve. $Re = 620, L_v/D_v = 0.14, \theta = 0^\circ.$	210-213
8.36-4 (1-4)	Filament of dye particles for a seated valve $Re = 620, L_v/D_v = 0.14, \theta = 45^\circ$	214-217
B-1	A composite grid-system	252

LIST OF TABLES

<u>TABLE</u>	<u>TITLE</u>	<u>PAGE</u> <u>NUMBER</u>
2.3-1	Constants in the k- ϵ turbulence model	39
4.5-1	Effect of grids on the U velocity	67
4.5-2	Computational details for flow through axisymmetrical expansions	70
5.3-1	Dimensions for flow through a seatless valve	89
5.5-1	The influence of grid number on the discharge coefficient	93
5.5-2	Computational details for flow through a seatless valve	94
6.3-1	Characteristics of the 21 cases for flow through a valve with an angled seat	112
6.5-1	Effect of grids in seat region on the velocity	117
6.5-2	Effect of grid in lift region on the velocity	117
6.5-3	Effect of grids on the discharge coefficient	119
6.6-1	The maximum recirculating mass flow rate and length of the reverse flow zone for three valve lift	130

<u>TABLE</u>	<u>TITLE</u>	<u>PAGE</u> <u>NUMBER</u>
6.6-2	Discharge coefficient for flow through a valve with 45° seat angle	135
6.6-3	Effect of the seat angle on the discharge coefficient	138
7.21-1	Dimensions of valves tested	148
7.3-1	Summary of the experimental conditions	154
8.1-1	Cases studied for simulation of experiments	179
8.22-1	Dimensions of the valve-port	182
8.23-1	Steady cases studied	183
8.23-2	Effect of grids on the velocity value at A	185
8.25-1	Measured and predicted the length of recirculation flow zone	188
8.25-2	The predicted and experimentally obtained positions of the vortex cores	196
8.32-1	Unsteady flow studied cases and grid employed	199
8.36-1	Measured and predicted position of the vortex centres	219

CHAPTER ONE - INTRODUCTION

1.1 The Problem

The present investigation is concerned with a study by a computer-based, numerical model and by experiments on a water model, of the characteristics of the flow region in which a poppet valve is placed to control the flow.

In order to facilitate the simulation, an axisymmetrical arrangement is chosen; this consists essentially of an idealized valve placed downstream of the joint between a short length of a circular pipe and a cylinder of a larger diameter, as shown in Figure 1.1-1. The investigation is further restricted to incompressible flows into the cylinder through constant valve-openings.

Two valve geometries are considered: the simple, seatless one of Figure 1.1-1; and one with an angled seat, as shown in Figure 1.1-2.

The influence of such parameters as valve lift, valve seat-angle and seat-width on the characteristics of the flow around, and downstream of, the valve are studied.

In the water experiments, only laminar flows with constant flow-rates are considered. The experimentally obtained flow patterns are compared with corresponding predictions. In addition, computations for turbulent air flows are made for a range of different parameters, especially for conditions for which suitable published experimental data exist.

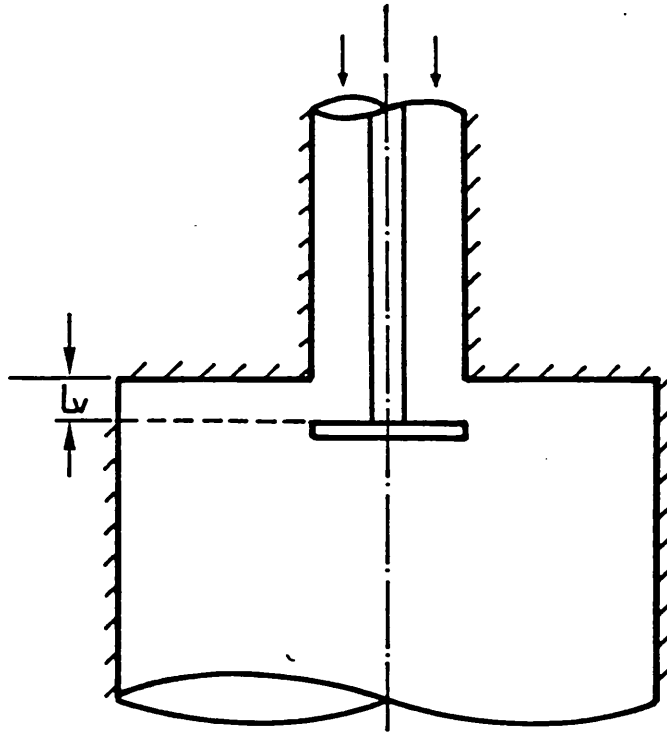


Figure 1.1-1 A seatless-valve assembly.

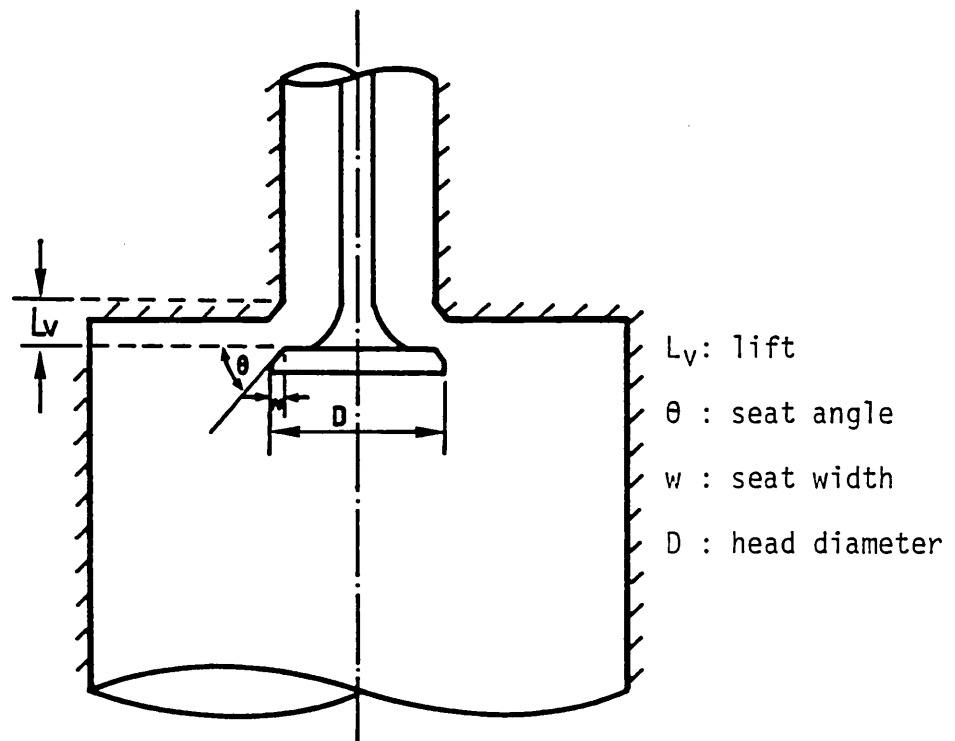


Figure 1.1-2 A seated-valve assembly.

1.2 Practical Relevance

A poppet valve is one of the seating-type valves in which the poppet, usually spherical or conical in shape, moves in a direction perpendicular to its seat. Its main function is to control the fluid flow into or out of a vessel such as a combustion chamber in an internal combustion engine.

The behaviour of the flow through the valves of an engine obviously influences the performance characteristics of the engine. In designing a more efficient engine it is thus necessary to understand fully the fundamental characteristics of the flow through valves.

A practical valve-port arrangement usually involves a complicated geometry, as shown in Figure 1.1-3; also, the flow through it is invariably unsteady and three-dimensional in nature. In the present study, simplifications to both the geometry and the dimensionality of the flow are introduced in order that computations and experiments can be carried out more easily and economically. However, since the simplified flow retains many of the important features of the practical problem, much insight into the characteristics of the real valve-flow can still be gained.

It is also hoped that experience obtained in the present study will help future simulations of the flow through a more practically realistic valve configuration.

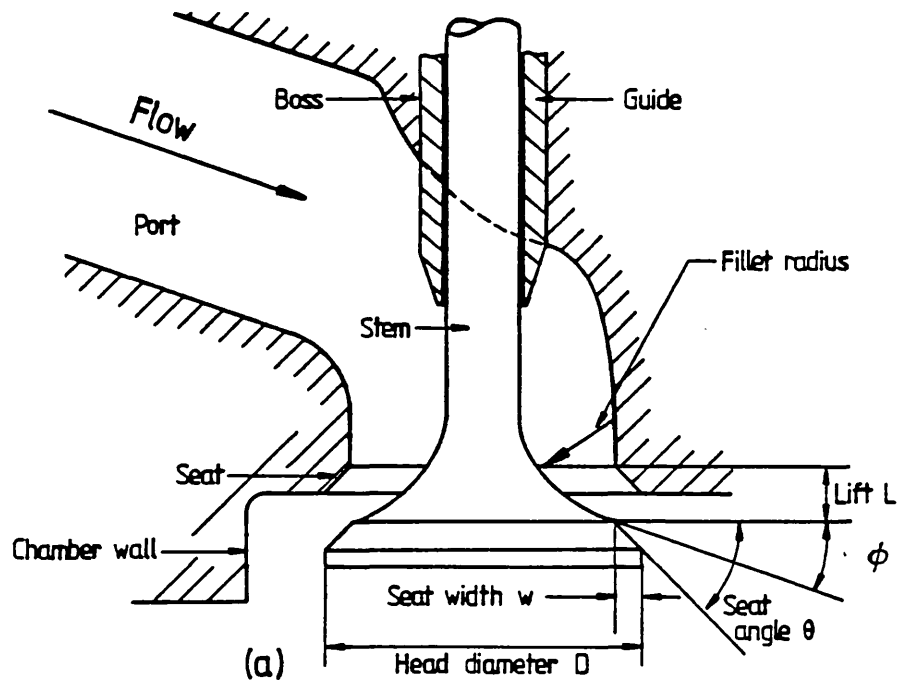


Figure 1.1-3 A typical inlet valve assembly.

1.3 Previous Work

The present review of the literature for works that are related to the inlet-valve problem under consideration is divided into two parts: experiments and theoretical computations.

1.31 Experiments

Probably because of its importance in influencing the performance of an engine, the Inlet valves of Internal combustion engines have received most attention. Nearly all the works reported have been concerned with global measurements such as the discharge coefficient, without attention to the detailed characteristics of flow. The experiments can be broadly classified into two groups.

In the first group are the 'static' tests, in which the air flow is measured through a stationary valve with the piston removed. The flow can thus be regarded as steady.

In the other group are the 'dynamic' tests. Two particular arrangements are often used: in one, the valve position is held fixed throughout the engine cycle; in the other simple movements of the valve are made in order to simulate the real engine conditions. All dynamic tests are necessarily unsteady in nature.

(1) Static Tests

Some experiments were carried out by Nutting and Lewis (1918) on the flow of air into a model combustion-chamber in the intake direction for several valve combinations. Only global quantities such as discharge

coefficient, pressure ratio etc were investigated. The tests covered a wide range of the valve lifts. Their results showed that the coefficient of discharge (see Appendix A) decreased with increasing lift.

The extensive work by Tanaka (1929) is notable for its thoroughness and its refinement in measuring techniques. Many tests were made with air flow through an inlet valve at various lifts. He was apparently the first to identify the following four distinct modes of flow over the full range of valve lifts.

In the first stage, at very low lifts, the flow remained attached to the wall of the valve passage, resulting in high values of the discharge coefficient. In the second mode, at slightly larger lifts, a vena contracta was found stemming from the underside curvature of the valve head, an abrupt fall in the discharge coefficient being associated with the occurrence of this mode. In the third stage, at intermediate lifts, a gradual improvement in the discharge coefficient was evident; and finally, at large lifts, any separation occurring was insignificant and consequently the discharge coefficient recovered as lift was increased.

Wood (1942) did some work similar to Tanaka's. Kastner, Walliam and White (1963) investigated both steady and unsteady flows, and confirmed qualitatively the four modes of flow noted by Tanaka. Their results for the unsteady experiment were found to be generally similar to those for the steady-state tests.

Annand (1969) performed a series of experiments on the steady flow through inlet valves of different geometries. His findings were generally similar to those of Tanaka (1929) and Kastner et al (1963). His results

of steady-flow tests were presented in the form of discharge coefficient as functions of the valve parameters (i.e. seat-width, -angle, and valve-lift). One of his conclusions was that the 60° seat-angle was generally inferior to the 30° or 45° one.

(2) Dynamic Tests

Kastner et al (1963) carried out some tests of unsteady flows through valves with a 45° seat angle. Ekchian and Hoult (1979), Fukutani and Watanabe (1982) investigated flow coefficients under conditions closer to those of a real engine than many others. However, they took global measurements only.

Bicen and Whitelaw (1983) used a laser-Doppler anemometer to measure the velocity distributions at the exit plane of an intake valve in a transparent model engine, under both steady and unsteady conditions and at various fixed valve-lifts. Qualitative agreement between the steady and unsteady flow field downstream of the valve was observed.

1.32 Theoretical Computations

A wide range of engine-simulation models have been comprehensively discussed in the literature by, for example, Reynolds (1978), Heywood (1980) and Witze (1981). They can be classified as being either zero-dimensional or the multi-dimensional.

Zero-dimensional models are usually thermodynamic in nature. They are formulated in integral form over space to remove any spatial variation within the cylinder; but temporal variations are allowed.

Multi-dimensional models employ partial differential equations. They represent the spatial variations of mass, momentum and energy in one-, two- or three dimensions and time, and thus account better for the flow processes.

Works on the zero-dimensional model have been surveyed in detail by Watkin (1978).

The present review on multi-dimensional calculations refers only to more recent studies. Nearly all the works which involve the inlet part predicted in-cylinder field values of the variables of interest, but none computed the flow field around the valve particularly. Watkin (1978), Romas (1980), Johns (1980) and Gosman (1982) etc treated the valve simply as a port, although they included other aspects of the flow important for piston/cylinder assembly studies.

1.4 Objectives of the Present Study

The objectives of this research were the following:

- (1) To use an existing computational procedure and the $k-\epsilon$ turbulence model to simulate the flow around an idealised poppet-valve, under steady and axisymmetric conditions:
- (2) To carry out parametric studies of the above flow and to compare the predicted values of the discharge coefficient with available experimental data:

- (3) To conduct a simple experimental investigation of water flow through a similar geometry but under laminar conditions; and
- (4) To predict quantitatively the flow structures observed in the experiments.

1.5 Present Contributions

Computations for the flow around a poppet valve under a range of different conditions have been made to provide information on the characteristics of such a flow. The computations were made using an existing PHOENICS computer code (Spalding, 1981). However, extensive problem-dependent inputs had to be provided in the form of SATELLITE, GROUND subroutine-attachments (See Appendix D,E,F) in order to simulate the particular complex geometries and the conditions considered.

Quantitative details of the flow velocity, recirculating mass-flow-rate, turbulence, etc. have been obtained and examined. In particular, values of the discharge coefficient for the valve as a function of the various parameters have been compared with the corresponding experimental data of Annand.

For the experiments, a simple water-model rig has been designed, constructed and tested. A Flow-visualization technique has been used to examine details of the flow observed. Global features of flow patterns have been recorded and compared with corresponding predictions, this provides a useful check on the accuracy and reliability of the computer-simulation technique. In the course of the experiments, some

unexpected phenomena have been noted.

1.6 Outline of the Thesis

The thesis is divided into nine chapters of which this introduction is the first.

In Chapter two is described the mathematical basis of the computational model, including the turbulence model used and the boundary conditions required to close the equation set.

Chapter three contains an 'outline of the numerical solution procedure embodied in PHOENICS.

The characteristics of the flow through an axisymmetric expansion are simulated, and predictions are compared with the available experimental data in chapter four.

In chapter five, computations for the flow of air through a seatless poppet valve for different valve lifts are presented and the results are examined.

Chapter six is concerned with flows of air through a valve with an angled seat, emphasis being placed on the effect of valve geometry on the discharge coefficient.

Chapter seven describes an experimental investigation, using a water-model, to study the flow around a simple valve assembly under laminar conditions.

Chapter eight describes the simulations of the experimental flows using the PHOENICS code. A particle tracking technique has been used to track the vortex formation in unsteady flows.

The last chapter, chapter nine, summarises the conclusions of this thesis and proposes some possible topics for future research work.

CHAPTER 2 - THE MATHEMATICAL PROBLEM

2.1 Introduction

In the analysis of any flow situation, the usual first step is to consider the general physical laws and the mathematical equations that describe the fluid dynamics of the flow. The present chapter provides the basic equations that govern laminar and turbulent flows, together with appropriate sets of the boundary conditions which are required adequately to define the problem.

The equations are given in the form applicable to flows usually described as 'elliptic', meaning that perturbation of conditions at any point in the flow can influence conditions at all other points. The phenomena studied in the present thesis are elliptic in character.

In section 2.2 are presented the conservation equations which govern the transport of mass, momentum and a general fluid property, ϕ , which, in the present context, can stand for either the turbulence kinetic energy or the turbulence dissipation rate.

In section 2.3 is given the turbulence model. It has not been developed or improved by the author. Therefore, it will not be discussed in great detail, but is presented here for completeness.

0

These equations do not by themselves constitute a closed set. The need for additional physical hypothesis to bring about a closure is discussed in section 2.4. and section 2.5 provides the boundary conditions and initial

conditions for the equations and finally, the closing remarks are presented in section 2.6.

2.2 Partial-Differential Equations

The time-dependent equations for the conservation of mass, momentum in the absence of external body forces can be expressed in the following tensorial form for newtonian fluids (Hinze, 1959).

Mass Conservation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho \bar{U}_i}{\partial x_i} = 0 \quad .(2.2-1)$$

Conservation of Momentum

$$\frac{\partial \rho \bar{U}_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho \bar{U}_i \bar{U}_j) = - \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) \right] \quad .(2.2-2)$$

where U_i and P represent respectively the velocities and static pressure, and ρ and μ are the fluid density and viscosity respectively. The symbol $[\bar{\cdot}]$ represents instantaneous value.

The above equations form the Navier-Stokes equations that predict the dynamic behaviour of laminar as well as turbulent flows. However, practical turbulent flows contain a cascade of eddy sizes that represent a wide range of time and length scales. Hence any numerical scheme using equations (2.2-1) and (2.2-2) for turbulent-flow simulations would require a grid fine enough to resolve even the smallest turbulent motions. This is not possible for the solution of practical problems as it is beyond the present computer capability.

Fortunately, for most engineering purposes, the problems are usually concerned only with the time-averaged effects of turbulence, even when the mean flow is unsteady. The 'statistical' representation of the turbulent flow by time-averaging is a useful approach. The time-averaged equations can be derived from the instantaneous equations presented in (2.2-1) and (2.2-2) by means of the following expressions:

$$\bar{U}_l = U_l + u_l' \quad , (2.2-3)$$

$$U_l = \frac{1}{2t_0} \int_{-t_0}^{t_0} \bar{U}_l \, dt \quad , (2.2-3a)$$

$$\frac{1}{2t_0} \int_{-t_0}^{t_0} u_l' \, dt = 0 \quad , (2.2-3b)$$

$$\bar{P} = P + p' \quad , (2.2-4)$$

$$P = \frac{1}{2t_0} \int_{-t_0}^{t_0} \bar{P}_l \, dt \quad , (2.2-4a)$$

$$\frac{1}{2t_0} \int_{-t_0}^{t_0} p_l' \, dt = 0 \quad , (2.2-4b)$$

where U_l , P are time-averaged values and,

u_l' and p' are their fluctuation components,

t_0 is a time interval.

Substituting relations (2.2-3) and (2.2-4) into the instantaneous flow equations (2.2-1) and (2.2-2) results in the following equations governing the mean motion of turbulent flow (Hinze 1959):

Mass Conservation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0 \quad .(2.2-5)$$

Momentum

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j) = - \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \overline{\rho u_i' u_j'} \right] \quad .(2.2-6)$$

In which the fluctuating velocity correlation tensor, $\overline{u_i' u_j'}$, known as the Reynolds stress, is defined as:

$$\overline{u_i' u_j'} = \frac{1}{2t_0} \int_{-t_0}^{t_0} u_i' u_j' dt \quad .(2.2-7)$$

This term physically represents the transfer of momentum by the turbulent motion. Due to its presence, the set of equations of motion presented above is not 'closed'. Thus additional relationships need to be developed to express $u_i' u_j'$ in terms of known or calculable variables. These efforts constitute the subject of turbulence modelling below (section 2.3). The equations for the conservation of scalar quantities such as turbulence energy and its dissipation rate etc. (though they are deduced from differing principles) all exhibit a similarity of form. Thus, they can be represented by the single equation for a general scalar property, ϕ , which is similar to equations (2.2-3 and 2.2-4). It is possible to define the time-mean ϕ and its fluctuating component ϕ' of ϕ such that:

$$\tilde{\phi} = \phi + \phi' \quad .(2.2-8)$$

$$\bar{\phi} = \frac{1}{2t_0} \int_{-t_0}^{t_0} \bar{\phi} \, dt \quad .(2.2-8a)$$

$$\frac{1}{2t_0} \int_{-t_0}^{t_0} \phi' \, dt = 0 \quad .(2.2-8b)$$

after substitution into and time-averaging the equation for the

instantaneous $\bar{\phi}$ to:

$$\frac{\partial \rho \bar{\phi}}{\partial t} + \frac{\partial}{\partial X_j} (\rho \bar{U}_j \bar{\phi}) = \frac{\partial}{\partial X_j} \left(\frac{\mu}{\sigma_\phi} \frac{\partial \bar{\phi}}{\partial X_j} - \overline{\rho u_j' \phi'} \right) + S\bar{\phi} \quad .(2.2-9)$$

where $S\bar{\phi}$ is the time-averaged source/sink of ϕ , and $\overline{\rho u_j' \phi'}$ is the turbulent transport term, which also requires modelling.

The equation set (2.2-5), (2.2-6) and (2.2-9), contains correlations of products of the time-averaging process: the turbulent (or Reynolds)

stresses, $-\overline{\rho u_i' u_j'}$, and turbulent fluxes, $-\overline{\rho u_i' \phi'}$. These are unknowns and, therefore, the equation set is not closed. The transport equations for these terms can again be derived from the Navier-Stokes equations but higher-order correlations will then be introduced; the closure cannot be achieved by resorting to equations for correlations of higher and higher order.

In order to close the system, the correlations are usually modelled in terms of quantities that can be calculated. It is described in the next section.

2.3 The "k-ε" Turbulence Model

The k-ε model is a two-equation model; it was first put forward by Harlow and Nakayama (1968). A full account of the model as used in the present work was given by Launder and Spalding (1973). A fundamental supposition of this model is that the turbulent quantities distribution in the flow field is adequately characterised by two transport equations for

- (1) k - kinetic energy of turbulence; and
- (2) ε - dissipation rate of turbulent kinetic energy.

They are related to the momentum equations by the concept of "eddy viscosity" which is also called the "turbulent viscosity". It relates the Reynolds stresses to the gradients of the mean velocities, and the turbulent flux to the gradients of the mean scalar quantities, by means of a turbulent viscosity, μ_t , as follows:

$$\overline{\rho u_i' u_j'} = - \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \rho k \quad .(2.3-1)$$

$$\overline{\rho u_i' \phi'} = - \frac{\mu_t}{\sigma_{\phi,t}} \frac{\partial \phi}{\partial x_j} \quad .(2.3-2)$$

where μ_t is the eddy or turbulent viscosity; δ_{ij} is the Kronecker delta; $\sigma_{\phi,t}$ is the turbulent Prandtl/Schmidt number, which is usually assumed to be invariant with direction and taken as a constant; and k, the time-averaged energy of the turbulent motion, is defined as:

$$k = \frac{1}{2} \overline{u_i'^2} \quad .(2.3-3)$$

The eddy viscosity approach to the modelling of turbulence is to relate the local turbulent viscosity, μ_t , to one or more properties of the turbulent flow. This viscosity is allowed to vary from one location in the flow domain to another; but at any one point, it is assumed to be isotropic. Here, it is evaluated from the local values of turbulent kinetic energy, k , and its rate of dissipation, ϵ , through the following relation:

$$\mu_t = C_\mu \rho k^2 / \epsilon \quad .(2.3-4)$$

where C_μ is an empirical constant.

An effective viscosity, represented by μ_{eff} , is defined as follows:

$$\mu_{eff} = \mu_l + \mu_t \quad .(2.3-5)$$

where μ_l is the laminar viscosity of the fluid. For high Reynolds number flows it is small compared to the turbulent one and may be neglected.

Both k and ϵ are determined by the following partial-differential equations which govern their transport:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial X_j} (\rho U_j k) - \frac{\partial}{\partial X_j} (\Gamma_k \frac{\partial k}{\partial X_j}) = S_k \quad .(2.3-6)$$

$$\text{and } \frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial}{\partial X_j} (\rho U_j \epsilon) - \frac{\partial}{\partial X_j} (\Gamma_\epsilon \frac{\partial \epsilon}{\partial X_j}) = S_\epsilon \quad .(2.3-7)$$

$$\text{where } S_k = G_k - \rho \epsilon \quad .(2.3-8)$$

$$S_\epsilon = \frac{\epsilon}{k} (C_1 G_k - C_2 \rho \epsilon) \quad .(2.3-9)$$

and C_1 and C_2 are empirical constants.

G_k is the rate of generation of turbulence energy which is given by the following expression:

$$G_k = \mu_t \left(\frac{\partial U_l}{\partial x_j} + \frac{\partial U_j}{\partial x_l} \right) \frac{\partial U_j}{\partial x_l} \quad .(2.3-10)$$

Γ_k and Γ_ϵ are the turbulent exchange coefficients for k and ϵ as given below:

$$\Gamma_k = \frac{\mu_t}{\sigma_k} \quad .(2.3-11)$$

$$\Gamma_\epsilon = \frac{\mu_t}{\sigma_\epsilon} \quad .(2.3-12)$$

where σ_k and σ_ϵ are turbulent Prandtl/Schmidt numbers.

The above equations involve the use of five constants; a consistent set of these is given in Table 2.3-1.

Table 2.3-1 Constants in the k - ϵ turbulence model (Launder and Spalding, 1974)

C_μ	C_1	C_2	σ_k	σ_ϵ
0.09	1.44	1.92	1.0	1.3

In the present study, the cylindrical coordinates are used, and the full set of partial-differential equations to be solved, in this coordinate, is summarised below:

Conservation of Mass:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho U)}{\partial Z} + \frac{1}{r} \frac{\partial (\rho v r)}{\partial r} = 0 \quad .(2.3-13)$$

Z - Momentum

$$\frac{\partial (\rho U)}{\partial t} + \frac{\partial}{\partial Z} (\rho U U) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho V U) - \frac{\partial}{\partial X} \left(\Gamma \frac{\partial U}{\partial X} \right) - \frac{1}{r} \frac{\partial}{\partial r} \left(r \Gamma \frac{\partial U}{\partial r} \right) = S_u. \quad .(2.3-14)$$

r - Momentum

$$\frac{\partial (\rho V)}{\partial t} + \frac{\partial}{\partial Z} (\rho U V) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho V V) - \frac{\partial}{\partial Z} \left(\Gamma \frac{\partial V}{\partial Z} \right) - \frac{1}{r} \frac{\partial}{\partial r} \left(r \Gamma \frac{\partial V}{\partial r} \right) = S_v \quad .(2.3-15)$$

k and ε Equations

$$\frac{\partial (\rho \phi)}{\partial t} + \frac{\partial}{\partial Z} (\rho U \phi) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho v \phi) - \frac{\partial}{\partial Z} \left(\Gamma_{\phi} \frac{\partial \phi}{\partial Z} \right) - \frac{1}{r} \frac{\partial}{\partial r} \left(r \Gamma_{\phi} \frac{\partial \phi}{\partial r} \right) = S_{\phi} \quad .(2.3-16)$$

where ϕ stands for either k or ε.

Γ_{ϕ} is the corresponding exchange coefficient.

S_{ϕ} is the source term for ϕ .

2.4 Auxiliary Relations

Before the equations above (2.3-13 to 2.3-16) can be solved, boundary

conditions (which will be listed in section 2.5) and such auxiliary relations as those for evaluating the density and viscosity, and the Reynolds stresses and fluxes (which has already modelled in section 2.3) are required.

(1) Density and Laminar Viscosity

In general, these properties may depend on the local state of the fluid (e.g. upon the pressure, temperature or composition). The density employed here is the constant. For all the calculations described in this thesis, uniform laminar viscosity is assumed.

(2) Reynolds-Stress and Turbulent-Flux Terms

A description has already been given in section 2.3 of the form of modelling adopted in the present work for the Reynolds stresses. Here, it is presented in order to cast the equations into the form in which they will be solved.

Many turbulence models have in common the central idea that, by analogy with the laminar stresses, the Reynolds stresses are related to the mean strain rate through the Boussinesq approximation, thus:

$$\overline{\rho u_i' u_j'} = -\mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \rho k \quad .(2.4-1)$$

Similarly, the turbulent transport term $\overline{\rho u_i' \phi'}$ may be expressed thus:

$$\overline{\rho u_i' \phi'} = -\frac{\mu_t}{\sigma_{\phi} \cdot t} \frac{\partial \phi}{\partial x_j} \quad .(2.4-2)$$

and k , the time-averaged energy of the turbulent motion is defined as:

$$k = \frac{\overline{u_i'^2}}{2} \quad , (2.4-3)$$

These three equations have already been given in section 2.3.

By doing this we have merely transferred attention from the Reynolds stresses to the turbulent viscosity, μ_t , which is related to k and ϵ via

$$\mu_t = C_\mu \rho k^2 / \epsilon \quad , (2.4-4)$$

Because of the similarity of form of the laminar and turbulent stress relations, it is convenient to combine them, and to work with an effective viscosity (μ_{eff}) defined as the sum of the laminar and turbulent component, thus:

$$\mu_{eff} = \mu_t + \mu_l \quad , (2.4-5)$$

2.5 Boundary and Initial Conditions

In addition to the equations presented above, the mathematical specification of the problem also requires information about the boundary and initial conditions. The former refers to a specification of values of all dependent variables at the edges of the flow region while the latter refers to their starting values at the field location.

The following types of physical boundary will be encountered in the

present study: axis of symmetry; inlet; outlet and walls. The treatment of these boundary conditions is as follows:

Axis

At the axis of symmetry the normal velocity is zero, and the normal gradient of all other quantities is zero.

Inlet, Outlet

The inlet and the far downstream outlet boundary conditions depend on the particular problem to be solved; they are specifically prescribed as will be shown in later chapters.

Walls

At a wall which is stationary and impervious, all components of velocity are zero, and the flux of species through the wall is also zero. However, as explained in the literature (Launder and Spalding, 1972a), close to the wall the effect of molecular viscosity becomes prominent because of the damping effect of the wall on turbulence and the properties of the flow vary steeply. The rigorous incorporation of these effects requires a prohibitively fine grid in the vicinity of a wall, and would be expensive. A practical approach avoids this difficulty by bridging the region near the wall with special functions which connect flow conditions at its outer edge to those at the wall. These 'wall-functions' are derived from experimental and analytical knowledge of the one-dimensional Couette flow which is assumed to exist very near the wall (Patankar and Spalding 1970, Launder and Spalding (1972, 1974)).

The velocity u_w parallel to the wall, in the region near the wall, is assumed to follow a logarithmic law expressed (Schlichting, 1968) as:

$$\frac{u_w}{u_\tau} = \frac{1}{\kappa} \ln(Ey^+) \quad .(2.5-1)$$

where u_τ is the 'friction velocity' given in terms of the wall shear stress, τ_w , by:

$$u_\tau = (\tau_w/\rho)^{1/2} \quad .(2.5-2)$$

$$\text{and } y^+ = \frac{\rho y_p u_\tau}{\mu} \quad .(2.5-3)$$

Here y_p is the normal distance of the position of u_w measured from the wall. κ and E are constants whose values are 0.4 and 9.0 respectively.

Near a wall, the transport equation (2.3-6) for the turbulent kinetic energy, k , reduces to one for a balance between its local production and dissipation (Launder and Spalding, 1972); thus, using equation (2.3-4),

$$\mu_{eff} \left(\frac{\partial u_w}{\partial y} \right)^2 = \rho \epsilon = \frac{C_\mu \rho^2 k^2}{\mu_{eff}} \quad .(2.5-4)$$

$$\text{and } \kappa = \frac{\tau_w}{\rho C_\mu^{1/2}} = \frac{u_\tau^2}{C_\mu^{1/2}} \quad .(2.5-5)$$

Using equations (2.5-1) and (2.5-2), the wall shear stress is given by:

$$\tau_w = \frac{\rho C_\mu^{1/4} k^{1/2} \kappa u_w}{\ln(Ey^+)} \quad .(2.5-6)$$

For a laminar flow in which $y^+ < 11.5$ (Launder and Spalding, 1972) the wall shear stress is simply given by:

$$\tau_w = \frac{\mu_l u_w}{y_p} \quad .(2.5-7)$$

In the near-wall region the production and dissipation rates of k must be modified to take account of the Couette flow existing along the wall. For the production rate of k , this is done by making use of the available

value of τ_w , thus the term $\mu_{eff} \left(\frac{\partial u_w}{\partial y} \right)^2$ is replaced by $\tau_w \frac{u_w}{y_p}$.

The dissipation rate of k is modified by means of equations (2.5-1), (2.5-2) and (2.5-3) and gives

$$\epsilon = \frac{C_\mu^{3/4} k^{3/2}}{k y_p} \quad .(2.5-8)$$

Initial Conditions

The initial conditions for the field are specified by assigning guessed values for steady calculations; or by using the field values obtained from the steady case for unsteady calculations.

In order to simplify the incorporation of these conditions into a finite-difference scheme, it is useful to note here that a field variable can be classified as either a specified value or zero normal flux.

2.6 Closure

In the present chapter, the equations which describe two-dimensional elliptic flows as deduced from the Navier-Stokes equations have been presented. The turbulence model employed for specifying the turbulent viscosity, μ_t , has been described, and Reynolds stresses and fluxes have been specified for 'closing' the equations. A standard form of the equations applicable to any generic fluid variable has been presented. The auxiliary information necessary to complete the mathematical specification has been outlined.

CHAPTER 3 - THE NUMERICAL SOLUTION PROCEDURE

3.1 Introduction

In the previous chapter, the differential equations which govern two-dimensional laminar and turbulent flows, along with the associated boundary conditions and auxiliary relations were set out. These equations are coupled and non-linear and are seldom amenable to analytical solution. The task in this chapter is to present the numerical procedure employed for the solution of these equations.

The equations are to be solved by means of the PHOENICS computer program (Spalding, 1981), which embodies a version of the finite-domain procedure developed by Patankar and Spalding (1972). Details of PHOENICS have been described by Spalding (1982); here only an outline is given, for the sake of completeness.

The solution procedure consists of the following major features:

- (i) The flow domain is divided into a finite number of discrete regions by constructing a staggered grid. This yields 'control volumes' or 'cells' for each flow variable.
- (ii) The partial-differential equations are transformed into finite-domain equations for each variable at each cell. This is accomplished by integrating the equations over the control volumes and using appropriate assumptions for the variation of the variables between grid nodes.

- (iii) The resulting finite-domain equations are non-linear and strongly coupled. Hence, an iterative solution procedure is required for their solution.
- (iv) The solution of the finite-domain equations can be influenced by the number of grid nodes employed. This number should be large enough for 'grid-independent' solutions to be obtained. The only way to determine the optimum number of grid to be used is to perform the calculation several times with differing numbers of grid nodes.

The remainder of this chapter consists of five parts in which the solution procedure is described. In section 3.2, the finite-domain equations are given, and the solution procedure is outlined in section 3.3. Detailed account of the auxiliary information is contained in section 3.4. Section 3.5 is the closure of this chapter.

3.2 Formulation of the Finite-Domain Equations

3.21 The Finite-Domain Grid

(1) Control Cells

The integration domain is filled by a set of non-overlapping cells which are constructed by orthogonal grid lines, as shown in Figure 3.21-1.

The cell-centre point p (called the grid node) is located at the geometrical centre of a cell. The neighbour nodes are referred to as points N, S, E and W (North, South, East, West); n, s, e and w are

used to denote the north, south, east and west faces of the cell respectively. The values of variables 'at' P, N, S, E and W are regarded as the representing values within the cells concerned while the values 'at' n, s, e, w are regarded as the representing values over the faces of a cell.

The grid may be non-uniformly spaced but once the grid density and dimensions have been chosen, they remain fixed during the solution process.

(2) Location of Flow Variables

Scalar variables such as pressure, density, exchange coefficient and kinetic energy of turbulence etc. are stored at the location P; the velocity components are stored at the cell-face locations, e, w, n, s, and are denoted by: U_e , U_w , U_n , U_s .

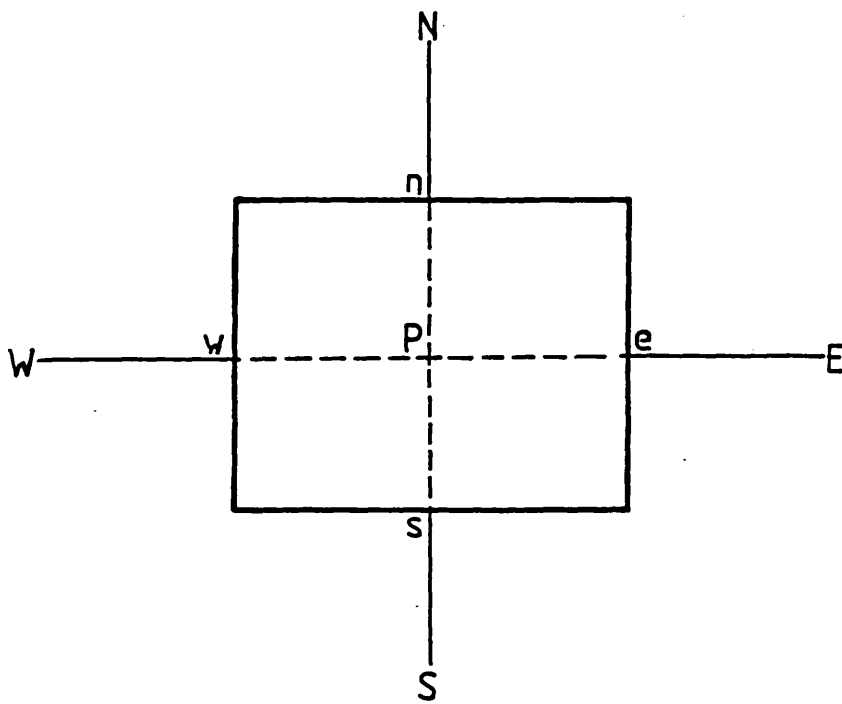


Figure 3.21-1 A two-dimensional control cell.

3.22 Derivation of the Finite-Domain Equations

The finite-domain equations for the scalar quantities are obtained by applying the conservation principle to a 'scalar cell' in which P is located (e.g. shaded region in Figure 3.21-2). Similarly equations for the velocities are obtained by applying the momentum-conservation principle to the 'velocity-cell', lying between C and D, A and B as shown as regions '2' and '3' in Figure 3.21-2 for velocities U and V respectively.

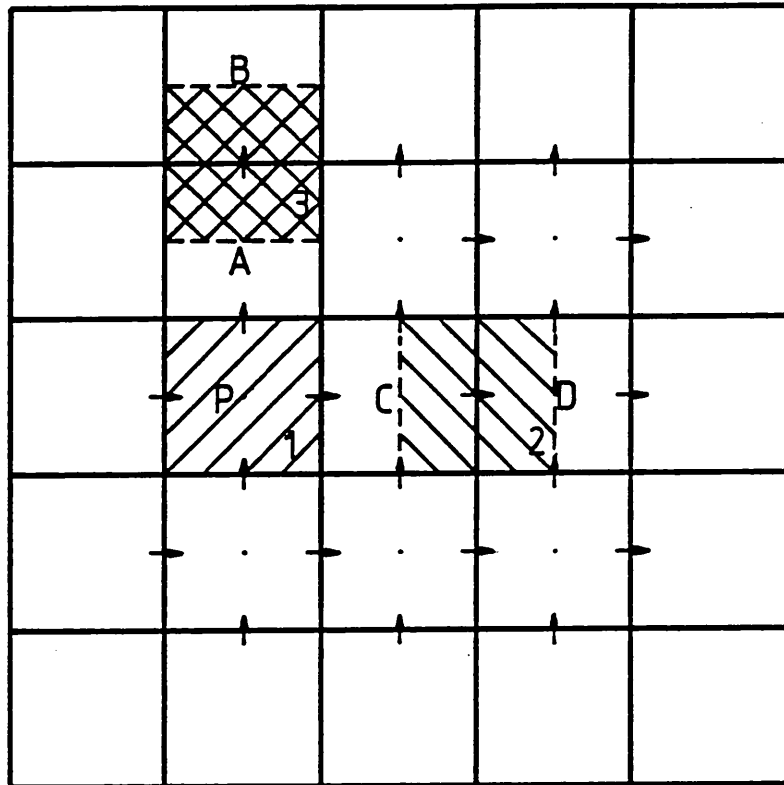
The derivation of the finite-domain equations has been described by Spalding (1982b). Therefore it is only briefly outlined here.. The equations are derived by integrating the differential transport equations over the control volumes that subdivide the flow domain. The integrations entail 'interpolation assumptions' which are commonly referred to as the 'fully-implicit' and 'upwind' formulations.

Integration of the general conservation equation (2.2-9) over the control cell P gives:

$$\iiint_V \frac{\partial(\rho\phi)}{\partial t} dV + \iint_A (\rho U\phi + J_\phi) \cdot dA = \iiint_V S_\phi dV \quad .(3.2-1)$$

Here V denotes the cell volume, A its surface area and J_ϕ the diffusion flux which is defined as:

$$J_\phi = -\Gamma_\phi \text{grad } \phi \quad .(3.2-2)$$



Symbol	Location	Control volume for
1	.	P, k, ϵ
2	→	U
3	↑	V

Figure 3.21-2 Grid showing locations of variables and their control volume.

Since variable ϕ is assumed to be uniformly distributed throughout the control cell, the transient term can be written as:

$$\iiint_V \frac{\partial(\rho\phi)}{\partial t} dv = V \frac{\partial(\rho\phi)}{\partial t} = \frac{V}{\delta t} (\rho_p\phi_p - \rho_{p-}\phi_{p-}) \quad .(3.2-3)$$

The subscript '-' refers to values prevailing at the start of the time interval. For the source term:

$$\iiint_V S_\phi dv = V S_\phi \quad .(3.2-4)$$

variables in the surface integral are presumed to be constant over each cell face; thus:

$$\iint_A (\rho U\phi + J_\phi) \cdot dA = \sum_{e.w.n.s} (\rho U\phi + J_\phi) \cdot A \quad .(3.2-5)$$

Equation (3.2-5) can be expressed in terms of the convection flux, C and diffusion flux, D, where:

$$C_l \phi = \rho u_l A_l \phi \quad .(3.2-6)$$

$$\text{and } D_l \phi = A_l \Gamma_l \phi / (X_l - X_p) \quad .(3.2-7)$$

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

The corresponding convection and diffusion coefficient (i.e. the C's and D's) are combined to form the coefficient of the finite-domain equations:

$$A = C + D \quad , (3.2-8)$$

(for details see Spalding and Rosten, 1981, and Spalding, 1982).

The general form of the 'finite-domain equations' (FDE's) is as follows:

$$A_P \phi_P = A_N \phi_N + A_S \phi_S + A_W \phi_W + A_E \phi_E + A_T \phi_T + S_U \quad , (3.2-9)$$

$$A_P = A_N + A_S + A_W + A_E + A_T + (-S_P) \quad , (3.2-10)$$

where $A_P, \dots, A_N$ etc. are coefficients; subscripts have the 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,

T grid node at earlier time.

S_U and S_P are appropriate source terms of ϕ for the cell,

after S_ϕ , which appears in equation 3.2-1, has been linearised as:

$$S_\phi = S_U - S_P \phi_P \quad , (3.2-11)$$

The formulation of all coefficients can be found in "Four lectures on the PHOENICS computer code" (Spalding, 1982).

3.23 Incorporation of Boundary Conditions Into the Finite-Domain

Equation

The boundary conditions required to specify the flow situations in the present problem have been discussed in Chapter 2. The appropriate methods of implementing them in the solution scheme are given below.

The values of the dependent variables must be specified at the boundaries before the finite-domain equations can be solved. These boundary conditions, which determine the uniqueness of the solution for a given flow situation, are usually of two types:

- (I) The value of the variable is prescribed at the boundary. This is generally accepted as the most stable of boundary conditions.
- (II) Another one, which is more commonly used, is the prescribed flux boundary (such as mass flux, momentum flux etc).

In the present study, both conditions are handled by way of the linearised source terms. The details of this approach can be found in Spalding (1982), and only a brief outline is described here.

The expression of the boundary condition in PHOENICS is a linear form. The total source is expressed as:

$$S = \dot{m} \phi_{BC} + C (\phi_{BC} - \phi_P) \quad , (3.4-1)$$

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

where

P_p is the pressure in cell p ;

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 (for details see Spalding, 1982).

3.3 The Solution of the Finite-Domain Equations

3.31 General Features of the Solution Method

The equations for the velocities, scalar property ϕ and pressure correction are expressed in the general form (3.2-9). Although they are linear in appearance, they are in fact non-linear, because the coefficients and sources are functions of ϕ 's. In addition, the momentum equations are coupled through the unknown pressure field. Thus, an iterative procedure must be employed for their solution.

In the present study, the finite-domain equations for U and V are solved in PHOENICS by the Jacobi point-by-point procedure. Most of the equation (scalar ϕ) sets are solved, slab-by-slab, by means of a slab-wise linear solver.

The SIMPLEST algorithm, which is developed from SIMPLE, is embodied in PHOENICS. The purpose is to solve the pressure-correction equation,

which takes the form of equation 3.2-9 and appropriate coefficients. It ensures that the continuity equation is satisfied, and the pressure values are calculated. The details of these have been described by Spalding (1982).

3.32 Sequence of the Operations in the Solution Procedure

The calculation sequence is carried out as follows:

- (i) The calculation starts at the slab (column in 2D case) of cells at the lowest x- location (i.e. near the west boundary). The pressure field is guessed.
- (ii) The convection fluxes, and all diffusive-flux coefficients, are constructed and stored at each main cell in the slab. (The coefficients of the finite-domain equation are computed and stored).
- (iii) The velocity components of U and V are solved for each control cell in the current slab, by the Jacobi point-by-point method.
- (iv) The finite-domain equations are set up for the remaining dependent variables (e.g. k and ϵ) and then solved.
- (v) The convection fluxes are recalculated, followed by the continuity error.
- (vi) The pressure-correction equation for each scalar cell is solved.

- (vii) The next task is to translate the pressure corrections into terms of corrections to the velocities, convections and pressures, which will cause the mass-conservation principle to prevail for all cells within the slab.
- (viii) Iteration may be performed at a slab (column in 2D case) by returning to step (i) and repeating. In a nearly-parabolic flow, much iteration is desirable, because few sweeps will be needed. In a strongly elliptic flow, few iterations are necessary, because the elliptic effects are better accounted for by sweeping through the solution domain.
- (ix) Attention is now shifted to the next slab, and the whole process is repeated. In this manner all slabs of the domain are visited to complete a 'sweep' of the integration domain. For a strongly elliptic flow, many sweeps may be needed to produce the required degree of convergence.

3.4 Control and Monitoring of the Solution Procedure

Convergence may be promoted in two ways, namely by use of:

(i) Inertial Under-Relaxation

Inertial under-relaxation may be applied to any ϕ variable. It adds the following term to the finite-domain equation:

$$S_{\phi r} = \lambda (\phi_p - \phi_{p,old}) \quad .(3.4-3)$$

where λ is a positive number. In PHOENICS:

$$\lambda = \frac{\rho \text{ Vol}}{DT_f} \quad .(3.4-4)$$

$S_{\phi r}$ is the term added and $\phi_{p,old}$ is the previously-existing value of ϕ at the cell in question. For large λ , this has the effect of keeping ϕ_p close to $\phi_{p,old}$. λ is often taken as a factor times the fluid mass in the cell:

$\frac{\rho \text{ Vol}}{DT_f}$ can be regarded as the reciprocal of a 'false time step'.

(II) 'Under-Relaxation' of the Pressure Correction

Under-relaxation of p' is a means of reducing the magnitude of the pressure correction:

$$p'_{p,new} = R_f p'_p \quad .(3.4-5)$$

where $p'_{p,new}$ is the new value of the pressure correction and R_f is the under-relaxation factor value lying between 0 and 1.

Convergence

It is important to identify whether or not the solution is converging, and if it is, whether it has sufficiently converged for practical purposes. The presence of systematic behaviour of ϕ 's between successive sweeps, (or iterations), with the difference diminishing, is one sign of convergence.

The convergence is also quantitatively monitored through the use of residuals. A residual of a variable ϕ at a node p , denoted by R_{ϕ} , is defined as:

$$R_{\phi} = A_p \phi_p - \sum A_L \phi_L - S_u \quad .(3.4-6)$$

where A_p is the denominator of equation (3.2-9). The simultaneously-existing residuals are defined as those present for each cell for each variable when the whole variable field has been frozen at the most-recently calculated value. To facilitate monitoring, the residuals are normalized by the product of a typical ϕ and the mass flow rate, $\dot{m}_b$, through the domain. For many purposes:

$$\sum_x \sum_y \frac{|R_{\phi}|}{\dot{m}_b \phi_t} \leq 0.05 \quad .(3.4-7)$$

may represent adequate accuracy; here ϕ_t is the 'typical' ϕ .

3.5 Closure

In this chapter, the finite-difference forms for the partial differential equations governing the generic fluid variable and the method of solution have been presented.

CHAPTER 4 - FLOW OF AIR THROUGH AN AXISYMMETRIC ENLARGEMENT

4.1 Introduction

The basic equations governing the transport of a fluid property, and the turbulence model employed, have been given in Chapter 2; and the method and means of their solution described in Chapter 3. In this chapter, the results of some computations based on these features are presented for an extensively-studied and well-documented flow situation, namely that of a turbulent flow through an axisymmetric enlargement. This problem was chosen primarily to test the validity of the turbulence model and wall functions incorporated in the PHOENICS code. However, as the flow bears some resemblance to the flow through a poppet valve, the main subject of the present thesis, it also served as a useful preliminary study to gain some insight into the characteristics of a recirculating flow.

The opportunity was also taken to test the use of "porosity" (see Section 4.5 below for explanation) in the PHOENICS code as a means of handling irregular boundaries like inclined walls; for it involved an enlargement with a conical transition piece.

Computations are presented in the present chapter for the flow through two axisymmetrical expansions: an abrupt one and one with a half angle of 45° . The predictions are compared with the appropriate experimental data of Chaturvedi (1963).

The chapter is divided into eight sections, of which this Introduction is the first. Section 4.2 describes the phenomena to be predicted. In

Section 4.3 the geometry and dimensions are given. The boundary conditions and the computational details are described in Sections 4.4 and 4.5. The predicted results of velocity, turbulence energy, streamlines and mean pressure are presented and compared with the measured data in Section 4.6. Section 4.7 contains discussion. The closure is in Section 4.8.

4.2 The Problem to be Predicted

When fluid flows steadily from a pipe into a coaxial one of a larger diameter, either through a sudden expansion or through one with a half-angle of 45° , a region of recirculating flow appears immediately downstream of the enlargement section. At a high Reynolds number (about 1.0×10^5 , based upon the diameter of the inlet pipe in the present study), the flow is turbulent. If the flow extends far enough, it reattaches to the wall and subsequently reaches a fully-developed state.

This study is concerned with the prediction of pressure, two velocity variables, and two turbulence variables for this particular flow.

4.3 Details of Geometry and Dimensions

Figure 4.3-1 shows the two geometries of axisymmetric expansions investigated here; these have also been used in the experiments by Chaturvedi.

The dimensions are as follows, the symbols being those shown in the figure:

$$D_o = 0.05\text{m} \quad .$$

$$D = 0.1\text{m} \quad .$$

$$L = 2\text{m} \quad .$$

$$l = 0.05\text{m} \quad .$$

$$\theta_1 = 90^\circ \quad .$$

$$\theta_2 = 45^\circ \quad .$$

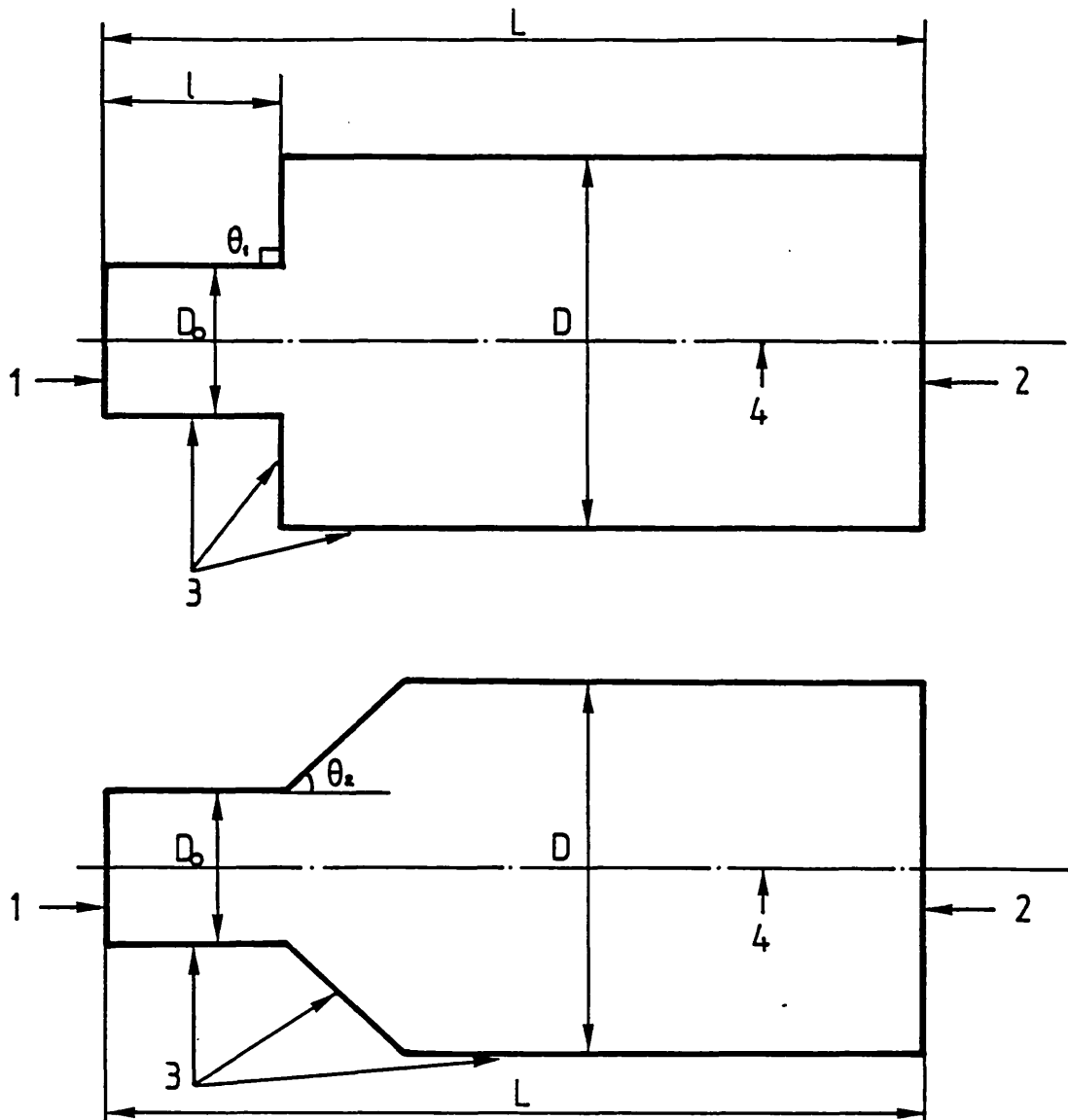


Figure 4.3-1 The geometries of the axisymmetric expansions with half angles of 90° (upper diagram) and 45° (Lower diagram)

(1) inlet

(2) outlet

(3) walls

(4) symmetry axis.

4.4 Boundary Conditions

The boundaries of the computational domain for these calculations comprise the symmetry axis, inlet, outlet and the walls, as shown in Figure 4.3-1.

Axis of Symmetry

Due to symmetry, it is appropriate to specify $V=0$ on this boundary; all gradients with respect to y are also zero.

Inlet Plane

The entering fluid is assumed to have a uniform axial velocity of 20 m/s, a turbulence kinetic energy of $1.2 \text{ m}^2/\text{s}^2$, and a turbulence-energy-dissipation rate is specified as $78.87 \text{ m}^2/\text{s}^2$.

Outlet Plane

A constant relative pressure, $P=0 \text{ kg/m}^2$, is specified on this boundary. No other conditions are needed, because no inflow occurs there.

Walls

Smooth-wall boundary conditions are used on all the walls; both v and w are set to zero on them. In the near-wall regions, wall functions are employed as described in Chapter 2.

4.5 Computational Details

4.5.1 Grid Selection

In order to examine the effects of grid size on the solution, systematic grid-refinements were carried out in both x- and y-directions. The effect of three different numbers of grid nodes on the dimensionless velocity (U/U_0) profile at the cross-section, $(x-l)/D_0 = 0.75$, for the abrupt expansion is shown in Table 4.5-1 and Figure 4.5-1. Figure 4.5-2 represents the influence of grid number on the velocity at point A in the high-shear region ($(x-l)/D_0 = 0.75$, $y/D_0 = 0.7$). It is seen that the solution obtained with the 20 non-uniform lateral-grid nodes and 34 in the axial direction can be regarded, on this evidence, as sufficiently grid-independent. The results presented for an abrupt expansion are based on the use of this particular grid, 20x34, as shown in Figure 4.5-3. Most of the nodes are located at the high-shear regions at the pipe junction and near the wall.

The same grid nodes are used for the expansion with a half angle of 45° .

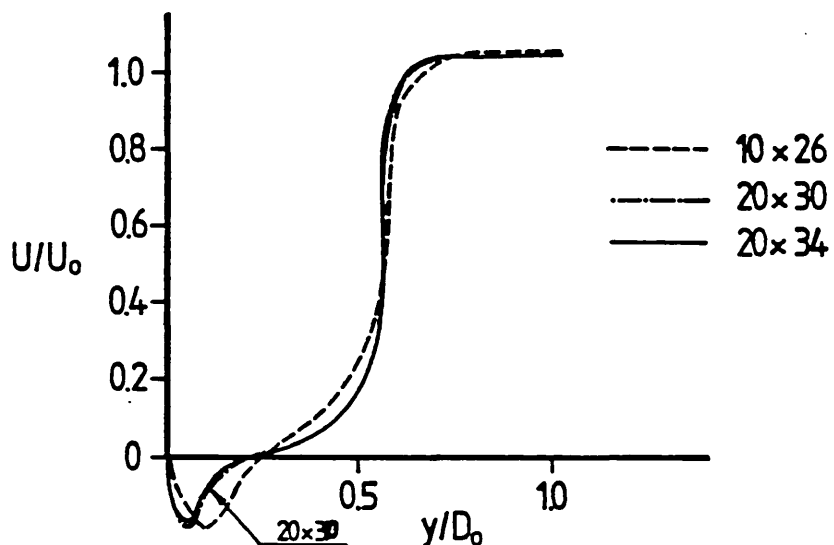


Figure 4.5-1: The velocity profile at 0.75 of $(x-l)/D_0$ for different sets of grid numbers.

Table 4.5-1: Effect of grid on the U velocity

	NY x NX	NY x NX	NY x NX
U/U_0	10 x 26	20 x 30	20 x 34
y ↑			
	-3.989	-1.847	-1.850 E-1
		-4.221	-4.224 E-2
		-1.969	-1.975 E-2
	-3.469	-3.569	-3.572 E-3
		9.765	9.795 E-3
	2.125	2.211	2.217 E-2
		3.511	3.516 E-2
	5.121	5.149	5.159 E-2
		8.659	8.661 E-2
	2.057	2.257	2.262 E-1
		8.603	8.609 E-1
	9.870	9.970	9.969 E-1
		9.991	9.909 E-1
	9.990	1.000	1.000 E-1
		1.000	1.000 E-1
	1.000	1.000	1.000 E-1
		1.000	1.000 E-1
	1.000	1.000	1.000 E-1
		1.000	1.000 E-1
	1.000	1.000	1.000 E-1

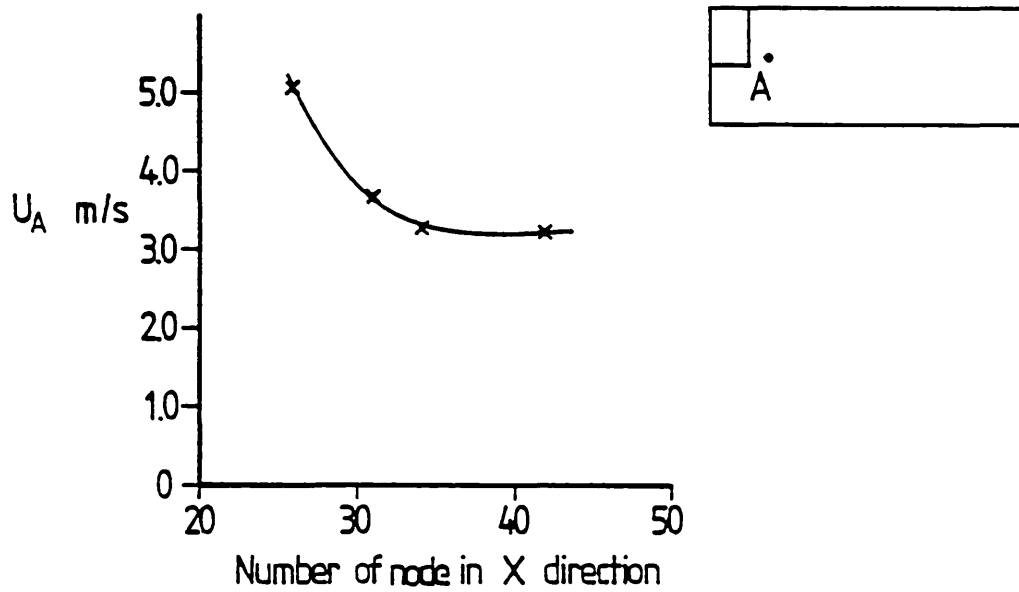


Figure 4.5-2 Influence of grid on the velocity at position

A, $(x-1)/b_0 = 0.75$, $y/D_0 = 0.7$.



Figure 4.5-3 The grid network for the abrupt expansion .

Use of Cell Porosity

In many processes which require computer modelling, the boundaries of the domain are irregular; or there are internal obstacles which are either totally or partially impervious to fluid flow. In the present procedure, these problems are handled by 'cell porosity'.

In this approach, every sub-domain is characterised by a set of fractions, normally in the range from 0.0 to 1.0. These fractions determine what proportion of each cell volume is available for occupancy by the fluid, and what proportion of each cell-face area can permit flow, by convection or diffusion, from one cell to its neighbour. Porosities of unity correspond to the situation of no blockage while that of zero corresponds to complete blockage. Values between zero and unity could represent a partial blockage, such as when an irregular flow-domain boundary passes across a cell, or a cell which is occupied by a distributed blockage (e.g. for the case with a 45° expansion angle, the volume porosity is 0.5). The details of setting these values are in the PHOENICS manual.

Other Computational Details

The calculation for the abrupt expansion requires 400 iteration sweeps of the domain for velocity and turbulence variables, with 10 iterations on the pressure-correction equation for each sweep; at the end of which the absolute volumetric error is reduced to 10^{-6} of the volumetric flux into the domain. These and other details of the computation are summarised in Table 4.5-2.

Table 4.5-2: Computational details for flow through axisymmetrical expansions

variable	value	
cases	abrupt expansion	expansion with 45° angle
grid NY x NX	20 x 34	20 x 34
type of grid spacing	non-uniform; mostly near walls and pipe-junction (see Fig 4.5-3)	non-uniform; mostly near walls and pipe-junction
iteration sweeps	400	500
inertial under- relaxation factor	0.1 (u) 0.1 (v) 10 ¹⁰ (other variables)	0.1 (u) 0.1 (v) 10 ¹⁰ (other variables)
net volumetric error volume flux	-3.180 x 10 ⁻⁶	-3.801 x 10 ⁻⁶
absolute volu- metric error volume flux	5.198 x 10 ⁻⁶	1.303 x 10 ⁻⁶
computer time	34 mins	57 mins

4.6 Presentation of Results

4.6-1 Abrupt Expansion

The predicted streamlines, velocity profiles, the turbulence energy and the mean pressure values are represented and compared with corresponding experimental data in the following paragraphs.

Streamlines

Streamlines are shown in Figure 4.6-1. The normalised stream function is defined as $\psi/D_0^2 U_0$; which is given by:

$$\frac{\psi}{D_0^2 U_0} = \int \left(\frac{\bar{U}}{U_0} \right) \left(\frac{X}{D_0} \right) \frac{dr}{D_0} \quad .(4.6-1)$$

$\bar{U}$ is the component of mean velocity in the axial direction.

D_0 is the diameter of the small pipe.

The separation streamline is shown by a dotted line.

Velocity Profile

Figure 4.6-2 represents the predicted distributions of the mean velocity at five different streamwise locations beyond the expansion. The mean velocity at each section is normalised by the reference velocity (U_{In}) of 20 m/s. and compared with Chaturvedi's data. The first three velocity profiles lie within the recirculation region, the values become negative near the cylinder wall indicating reverse flow there. The recovery of the

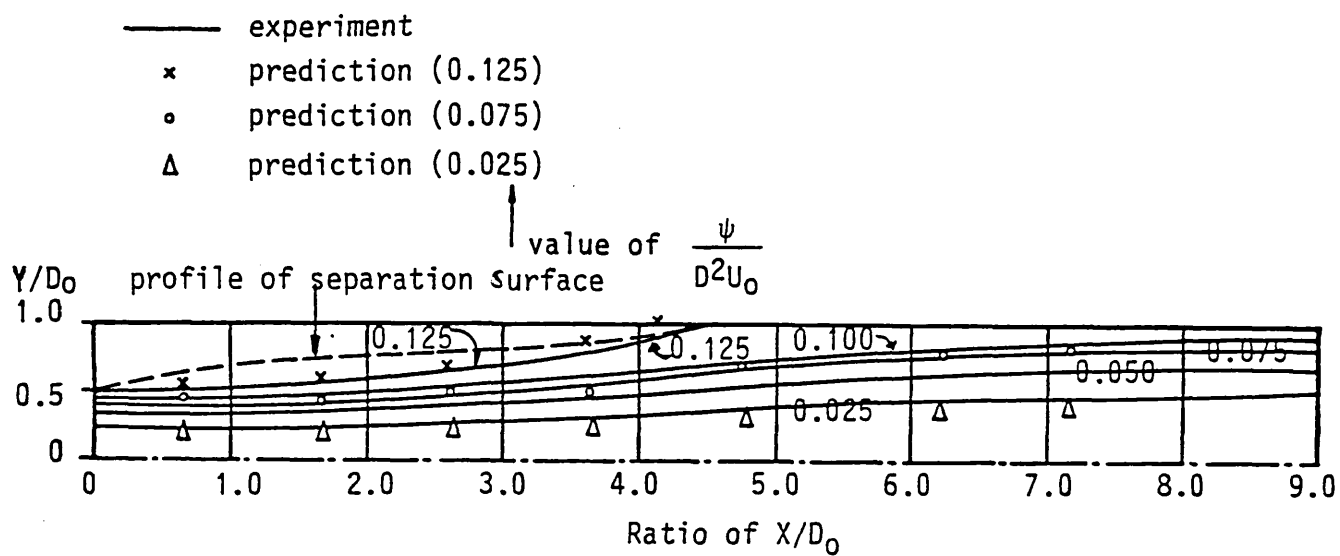


Figure 4.6-1 Pattern of streamlines, abrupt expansion.

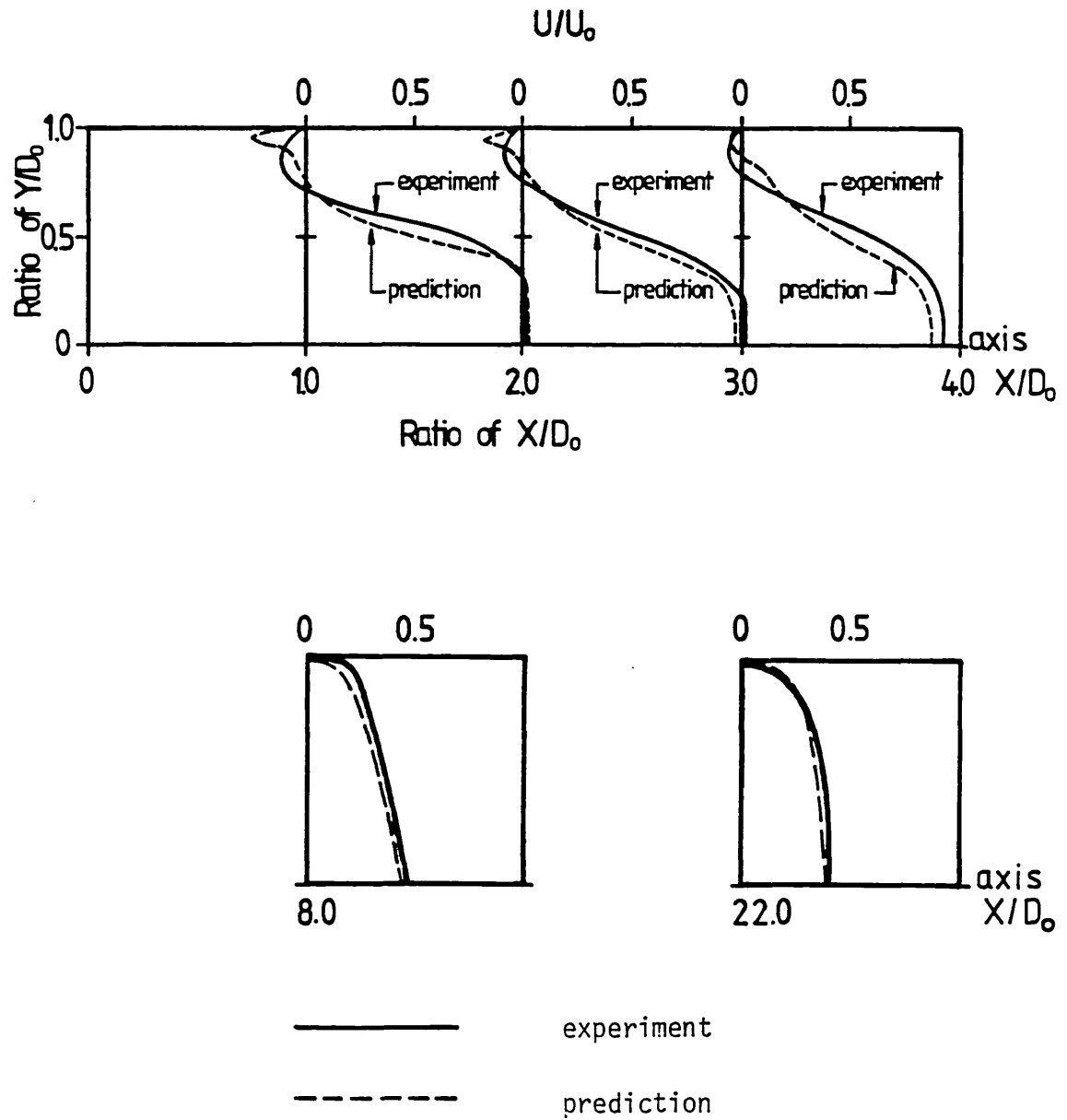


Figure 4.6-2 The mean velocity at different cross section,
abrupt expansion.

flow structure beyond the recirculation zone seems better simulated as shown in the velocity profile at $X/D_0 = 22$ in Figure 4.6-2

Turbulence Energy

In Figure 4.6-3 is shown the contour plot of the non-dimensional

turbulence energy in the flow field, (ie value of $\frac{\overline{\rho u'^2}}{\rho U_0^2}$), where

u' is the components of velocity-fluctuation in the x-direction about mean, and is obtained from:

$$u' = 2k^{1/2}/3 \quad .(4.6-2)$$

U_0 is the mean velocity at the inlet.

- experiment
- x prediction (0.02)
- o prediction (0.04)
- Δ prediction (0.06)

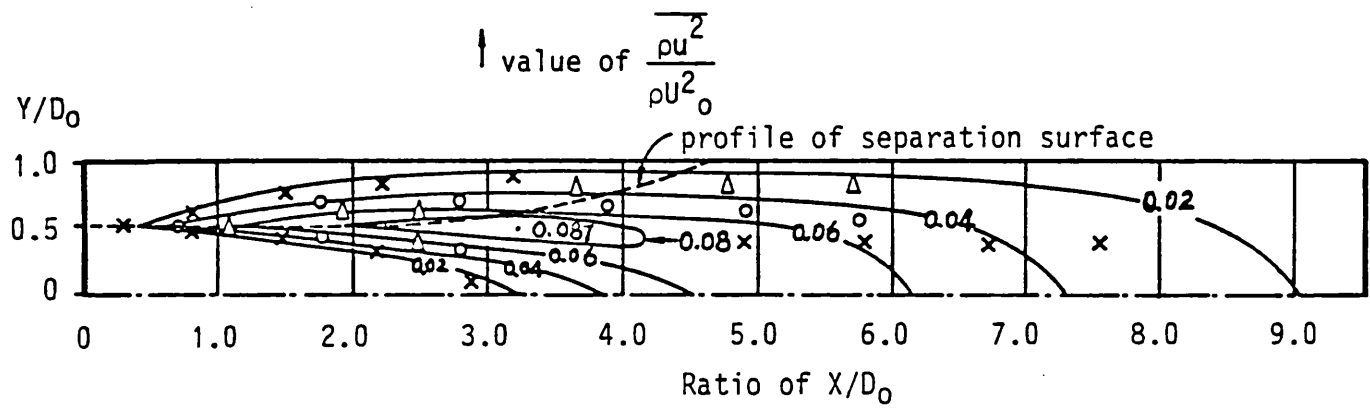


Figure 4.6-3 Spatial distribution of turbulence energy,
abrupt expansion.

Pressure Field

In Figure 4.6-4 is shown the spatial distribution of the mean pressure downstream of the expansion. For comparison with Chaturvedi's experimental data which was only available for the separation region immediately downstream of the step, the predicted results are plotted with the difference, P_d , between the local pressure and the one at the pipe junction, i.e. $-6.807 \text{ E}+1$, and normalised by $\rho U_0^2/2$. Figure 4.6-4 (A) shows the predicted radial distribution of mean pressure at six streamwise locations. The mean pressure is defined in the same way as in Figure 4.6-4.

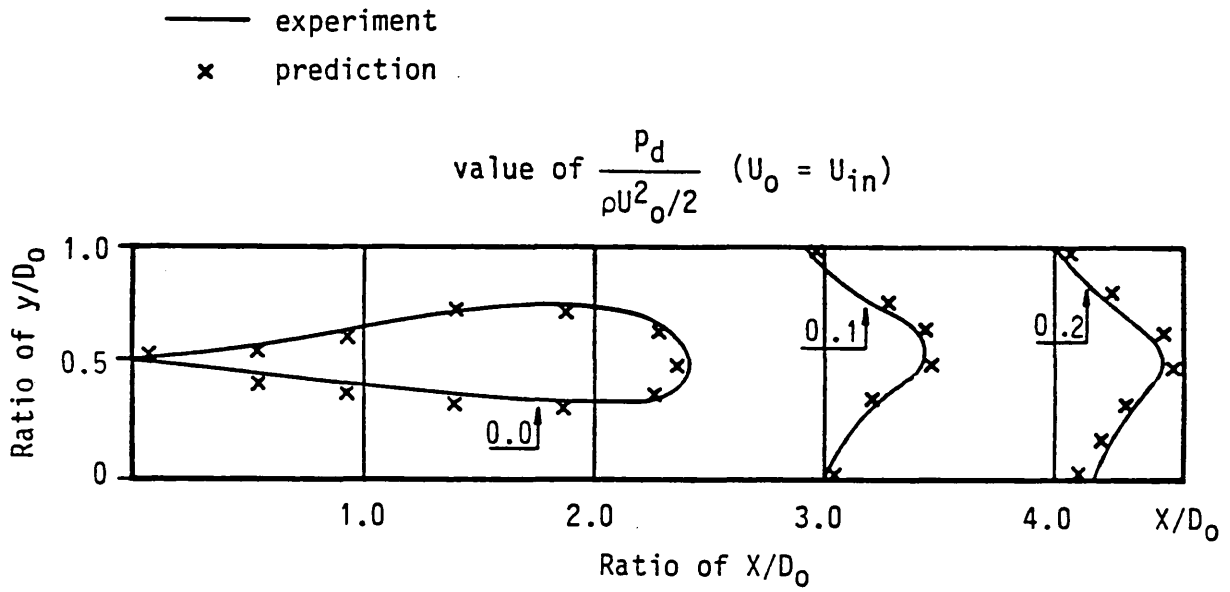


Figure 4.6-4: Spatial distribution of mean pressure, abrupt expansion.

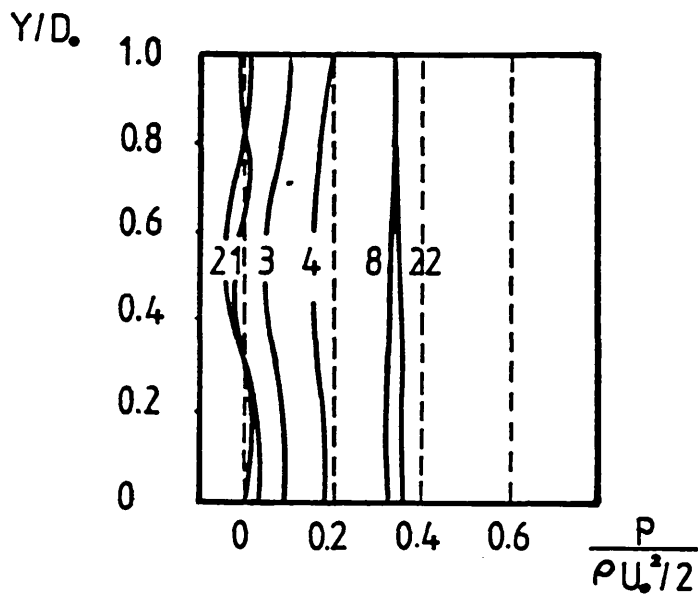


Figure 4.6-4 (A): Radial distribution of mean pressure at different streamwise locations.

4.62 Expansion with a Half Angle of 45°

This particular exercise has been carried out to test the porosity approach employed in PHOENICS. Only the streamlines, velocity profile and pressure distribution are predicted and compared with the experiment.

Figure 4.6-5 shows the streamlines. The stream function is defined in formula 4.6-1. The trend is similar to the abrupt expansion. In Figure 4.6-6 is shown the calculated radial profiles of the mean axial velocity at the same locations as in Figure 4.6-2. No significant difference between the abrupt expansion and the expansion with a half angle of

45° can be found. The spatial distribution of the mean pressure,

$\frac{P_d}{\rho U_0^2/2}$, which is similar to the abrupt expansion case, is

presented in Figure 4.6-7.

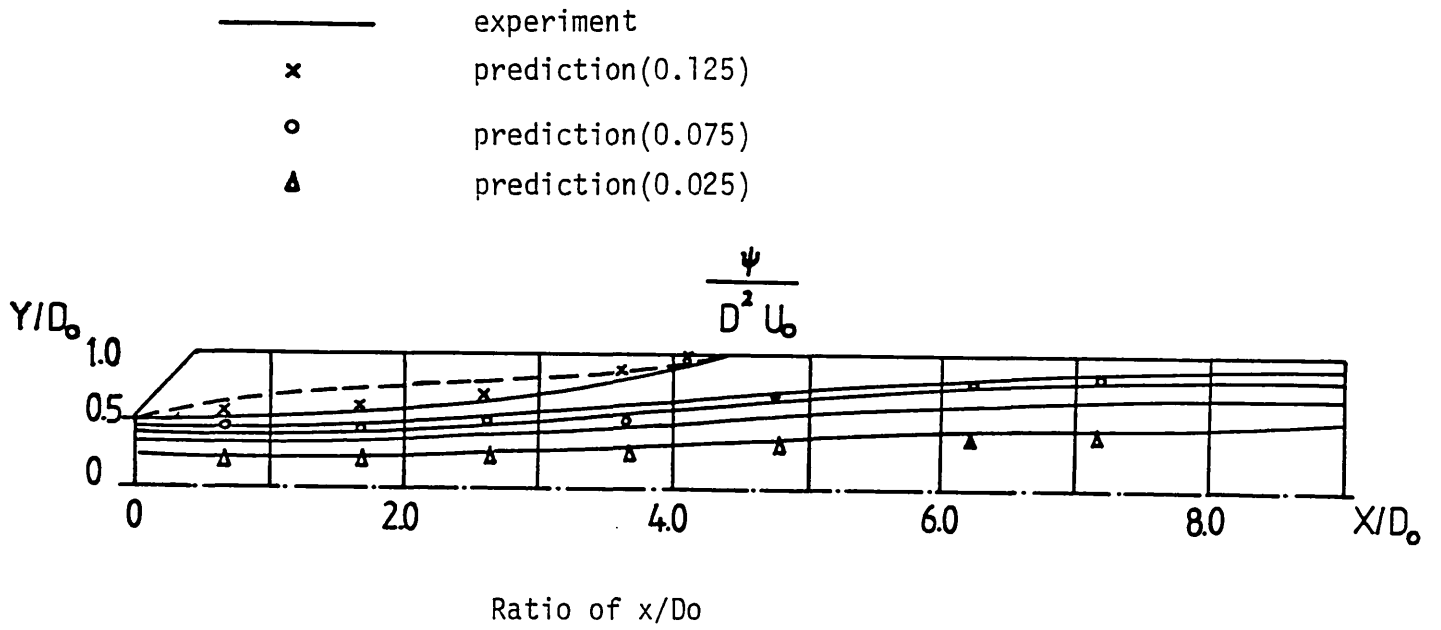


Figure 4.6-5 Pattern of streamlines, 45° expansion.

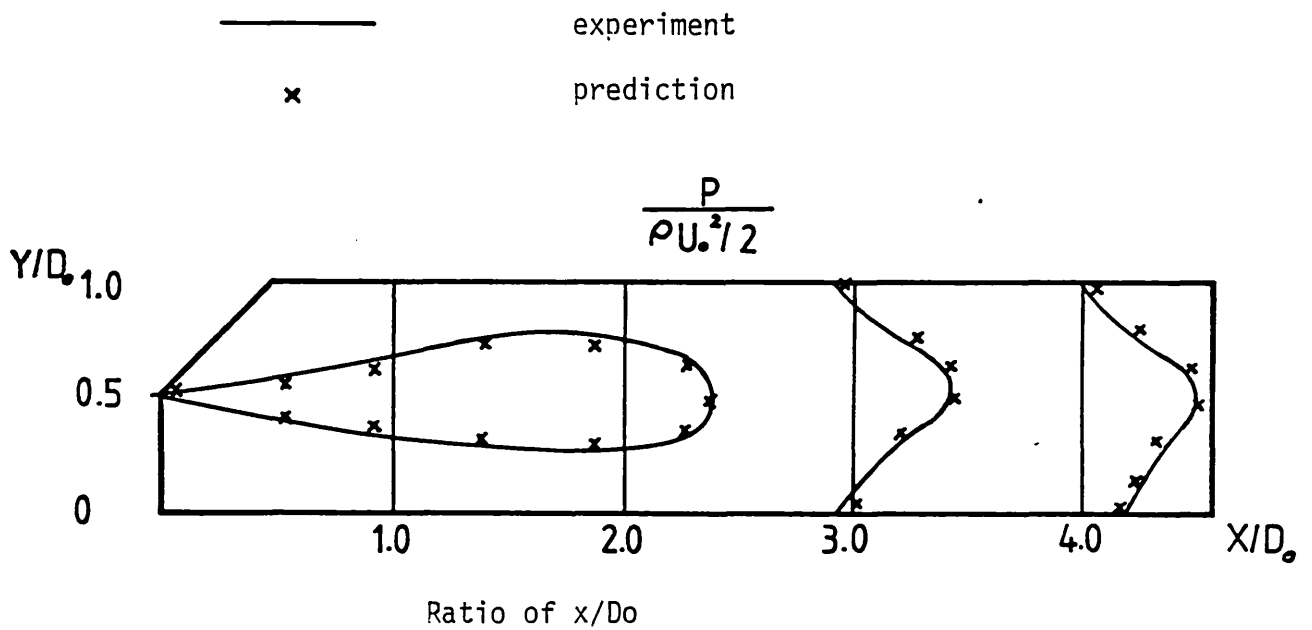


Figure 4.6-7 Spatial distribution of mean pressure,
45° expansion.

4.7 Discussion

The pattern of streamlines shown in Figure 4.6-1 may be considered generally satisfactory in comparison with Chaturvedi's experimental data. However, it should also be noted that the length of the recirculation zone, which is defined as the axial length from the step to the position where the mean streamline is reanchored to the wall, is slightly underpredicted.

The velocity distribution, as shown in Figure 4.6-2, is generally well predicted with the $k-\epsilon$ model. However, it also reveals that in the first two profiles, the magnitude of the predicted reverse flow velocity near the cylinder wall is less than the measured value, the maximum discrepancy is about 7%. The predictions agree well with the measurements beyond the recirculation zone, the maximum discrepancy being less than 2%.

Figure 4.6-3 demonstrates that the maximum turbulence energy occurs in the high-shear regions away from the walls. It is also seen that the predictions and measurements display similar trends. However, the agreement is reasonably good up to where $X/D_0 = 4.0$, but after that, the discrepancy between the predictions and experiments becomes larger (the maximum discrepancy is about 15%).

The comparison of the predicted mean pressure with Chaturvedi's data is shown in Figure 4.6-4. The agreement is reasonably satisfactory. The predicted radial distribution of mean pressure at different streamwise locations shown in Figure 4.6-4 (A) displays that the variation of the pressure in the axial direction is appreciable, and relatively small in the radial direction. It has also revealed that the pressure variation in the

radial direction at the separation region appears to be related to the turbulence characteristics: the higher the intensity of the turbulence, the lower the pressure of the mean value over the section.

The distributions of mean velocity, streamlines, turbulence energy, and pressure presented in Figures 4.6-1 to 4.6-4 so far give a detailed picture of the general flow characteristics of abrupt axisymmetric expansion. The feature is that the expansion generates high shearing rates in regions removed from the immediate vicinity of the pipe wall. The upstream boundary layer separates at the sharp corner forming a free-shear layer. In this region the turbulence energy generation rate will be large, while dissipation rates will be low because, away from the wall, large-scale motions predominate. The dividing streamline through the first half of the separated flow region is only slightly curved, and the shear layer thin enough not to be affected by the presence of the wall. Later, however, the separated shear layer curves sharply downwards in the reattachment zone and impinges on the wall. Part of the shear-layer fluid is deflected upstream into the recirculating flow by a strong adverse pressure gradient. The shear layer is subjected to the effects of stabilising curvature, adverse pressure gradient, and strong interaction with the wall in the reattachment zone. The recirculating flow region below the shear layer cannot be characterised as a dead fluid zone. The maximum backflow velocity is over 20% of the freestream velocity, the calculation values are even larger. Downstream of reattachment, the Reynolds stresses continue to decay rapidly for a distance of several small pipe diameters. Simultaneously, a new sub-boundary layer begins to grow up through the reattached shear layer.

The streamline plot, mean velocity and pressure distribution for the

expansion with a half angle of 45° , as shown in Figures 4.6-5 to 4.6-7, are compared with the measured data. Agreement has been shown to be reasonably good. It might therefore be inferred that the porosity method employed is adequate for the simulation of an inclined wall.

4.8 Closure

In this chapter, the $k-\epsilon$ turbulence model is applied to flow through two kinds of axi-symmetric expansion, namely abrupt expansion and expansion with a half angle of 45° . The porosity approach is used for the inclined wall boundary. The predictions have been compared with established experimental data and agreements have been found to be generally fair. However, it has also been revealed that the predicted turbulence energy distribution agrees only with the experiment up to $X/D_0 = 4$, after which the discrepancy increases. A reasonable conclusion might be that the numerical procedure and the turbulence model employed are realistic ones, and the porosity method is adequate for the simulation of an inclined wall.

CHAPTER 5 - AIR FLOW THROUGH A SEATLESS POPPET VALVE

5.1 Introduction

In the previous chapter the simulation of flow through axisymmetric expansion was presented. This chapter describes the simulation of valve flow.

A poppet valve is one of the seating-type valves in which the poppet, usually spherical or conical in shape, moves in a direction perpendicular to its seat. It is generally applied as a means of shutting off and as such it has been used for a long time. The geometry of a seatless valve, as shown in Figure 5.1-1, is simpler than that of a valve with an angled seat and is therefore chosen for a preliminary study to obtain some insight into the characteristics of a valve flow.

In a real engine, the valve-opening area varies with time. At any instant in the process, the flow rate through the open area is determined by the difference between the pressure in the cylinder and that in the ducting immediately upstream; these pressures also vary continuously. Few fairly comprehensive tests for the measurements with time-varying variables (Fukutani (1982), Tabaczynski (1984)) have been reported recently. However, many experiments (Wood (1967), Annand (1969), Bicen (1983) etc) have been carried out under steady conditions, which could provide an insight into valve flow with considerable economy and convenience. The available experimental data are usually restricted to simple measurements of global parameters, for example the discharge coefficient.

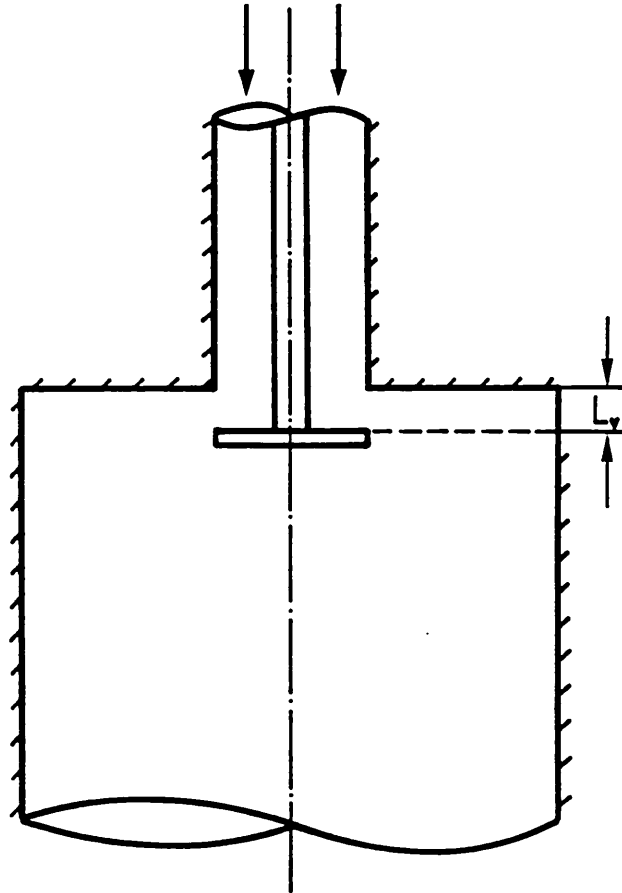


Figure 5.1-1 Inlet seatless valve assembly.

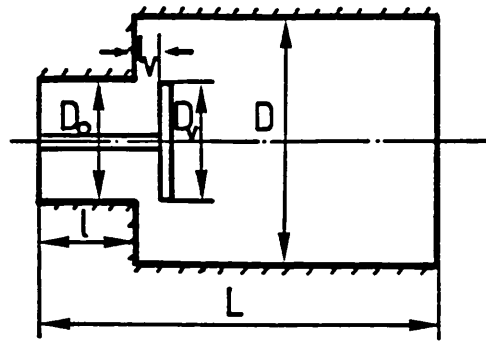
In the present chapter, the steady turbulent flows through a seatless valve with various valve lifts are simulated. The calculation is restricted to a simpler-than-practical geometry, ie an idealised, axisymmetrical, isolated seatless valve in a straight parallel port. It was tested up to a dimensionless lift (L_v/D_v) of 0.3, a range which is commonly used in practice. Two kinds of flow situations are studied numerically: one for flow through a valve into a cylinder and the other for flow to the open atmosphere, the latter corresponding to Annand's experiment. The predicted discharge coefficients for the two situations are compared with Annand's experimental data. The predictions also yield the flow field (which cannot be provided with a simple experimental apparatus), some details of which are plotted as centreline and radial profiles and contours.

5.2 Phenomena to be Predicted

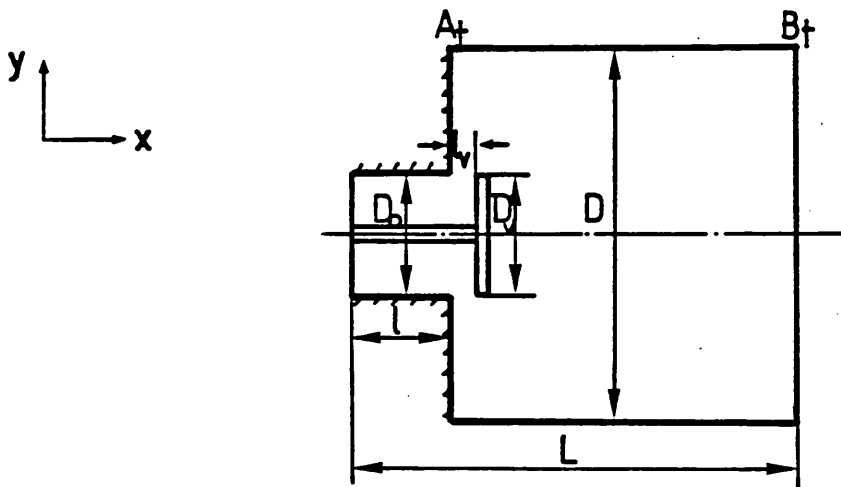
Much less is known of the details of the flow through a valve than that through an axisymmetric expansion as described in Chapter 4. It has been observed that in the former, one vortex ring occurs downstream of the step; in the latter, however, there is a second vortex ring appearing just downstream of the valve, and the position of the valve does influence the discharge coefficient. These phenomena are studied.

5.3 Geometry and Dimensions

Two geometries have been considered. They are shown in Figures 5.3-1 (A) and (B). The difference between the two cases is that in the first, a flow discharges into a large diameter cylinder; while in the second case the flow discharges into the atmosphere.



(a) Flow into a cylinder



$A_f B_f$ is the free boundary

(b) Flow into the atmosphere

Figure 5.3-1 (a) and (b) Geometries of the seatless-valve opening.

The valve dimensions for both cases are the same as those used by Annand:

valve diameter 0.045 m
valve stem diameter 0.0095 m

The dimensions of the valve assembly used in the numerical simulation cases are given in Table 5.3-1. Symbols used are the same as those in Figure 5.3-1.

Table 5.3-1: Dimensions of two cases

	flow into a cylinder case A	flow into the atmosphere case B
valve diameter D_v	0.045 m	0.045 m
cylinder diameter D	0.09 m	
calculation domain width D_F		0.8 m
calculation domain length L	0.5 m	0.5 m
lift: L_v/D_v		
(1)	0.053	0.053
(2)	0.1	0.1
(3)	0.15	0.15
(4)	0.2	0.2
(5)	0.25	0.25
(6)	0.3	0.3

5.4 Boundary Conditions

The boundaries of the computational domain are shown in Figure 5.3-1 and boundary conditions are specified as follows:

Axis of Symmetry

At the centre-line, the normal velocity, v , is zero and all normal gradients of other variables are zero.

Inlet Plane

At the inlet, a uniform velocity profile of 15 m/s is specified; a uniform turbulence kinetic energy of $0.675 \text{ m}^2/\text{s}^2$ and turbulence-energy-dissipation rate of $1.664 \text{ m}^2/\text{s}^3$ are prescribed.

Outlet Plane

A fixed constant pressure, $P = 0 \text{ kg/m}^2$, is specified on this boundary. As the fluid is assumed incompressible, computed pressures are relative to this value.

Walls

The regions near all walls, including the cylinder wall, the duct wall, and valve-face and -stem, are treated by use of the wall function described in section 2.

Free-Boundary

For case B, (flow through a valve enters the atmosphere), $A_f B_f$ is the free-boundary where a uniform constant pressure is assumed.

5.5 Computational Details

5.51 Grid-Dependence Study

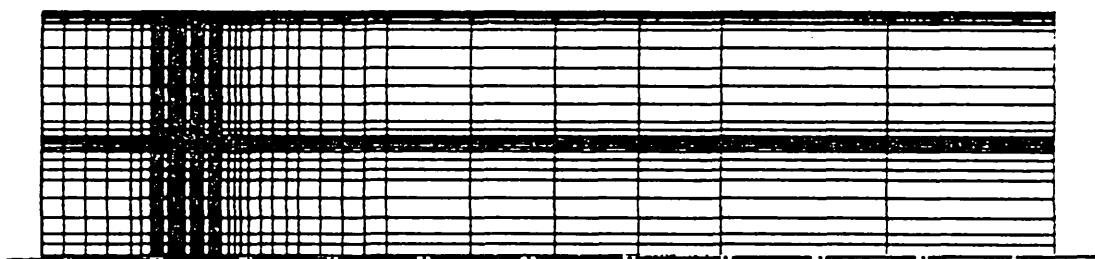
The calculation domain is shown in Figure 5.5-1. Also shown is the grid network used. The grid distributed non-uniformly over the flow domain with a large number of nodes in the vicinity of the valve and the cylinder wall.

The number of grids in y-direction is the same in Chapter 4. The influence of the axial grid number on the discharge coefficient is studied. As the region upstream of the valve is more important for the present problem, the effect of the grid number in this region (A) on the discharge coefficient for Case A, $L_v/D_i = 0.15$, is shown in Figure 5.5-2. The influence of the total grid number on the discharge coefficient for the same case is represented in Table 5.5-1.

It can be concluded, by reading the results in the table, that 20 x 34 grid-nodes provide sufficient 'grid-independence'.



a



b

Figure 5.5-1 Grid distribution of the calculation domain

(a) for the whole domain;

(b) part of the domain around the valve.

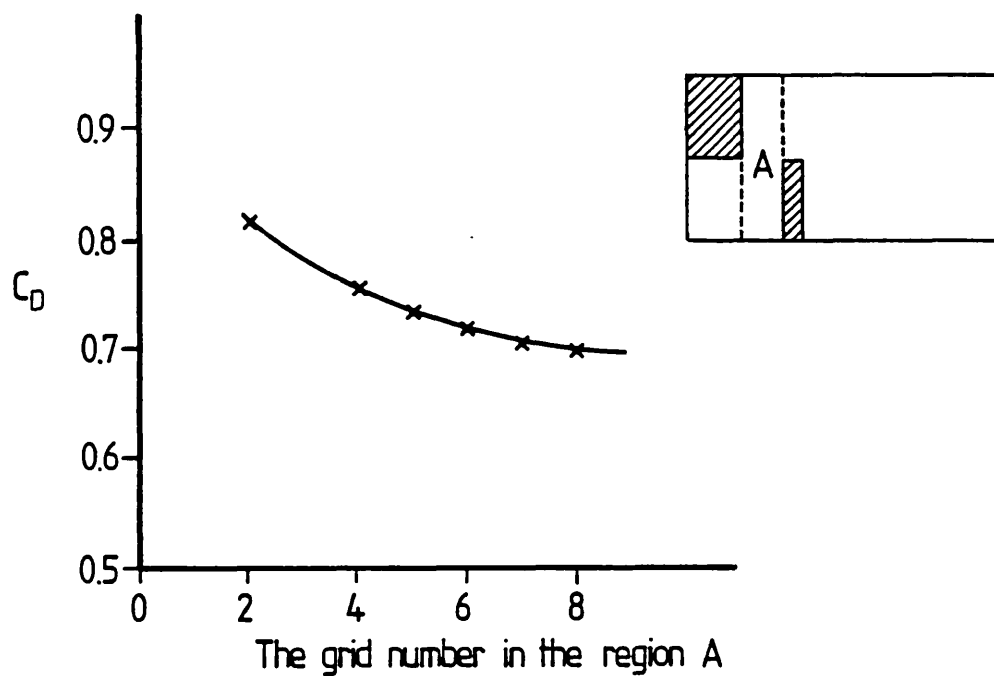


Figure 5.5-2 The influence of Grid number in the region A on the discharge coefficient

Table 5.5-1 The effect of the nodes in region A on the C_D .

grid No. in region A	2	4	5	6	7	8
C_D	0.8327	0.7612	0.7427	0.7281	0.7173	0.7162

5.52 Other Computational Details

Table 5.5-2 summarises the grid employed and other computational details for the same case on grid-dependence. The 'Net' volumetric error gives the sum of the volumetric imbalance of each cell: while the 'absolute' error gives the sum of the absolute volumetric imbalance of each cell. Of the two, the latter is considered to be more important for the former could be small by cancellation of large positive and negative errors. In Table 5.5-2 below, each of these errors has been non-dimensionalised by the volumetric flux, ie $1.0E-3$.

Table 5.5-2 Computational details for flow through a seatless valve with uniform properties

Variable	value
grid NYXNX	20 x 34
type of grid spacing	non-uniform, mostly near the valve and the cylinder walls
number of sweeps	600
number of iterations on v and u p'	1 10
inertial under-relaxation factor for u v k ε	0.1 0.0001 0.01 0.01
non-dimensionalised net volumetric error	4.185×10^{-5}
non-dimensionalised absolute volumetric error	7.261×10^{-5}

5.6 Presentation of the Main Results

The results of the main computations are considered under three headings: profiles of mean velocity and wall pressure; profiles of the turbulence kinetic energy and intensity; and the discharge coefficient.

Profiles of Mean Velocity and Wall Pressure

In Figure 5.6-1 is shown the predicted radial profiles of axial mean velocity at various streamwise locations upstream and downstream of the valve for $L_v/D_v = 0.15$ (Case A). The mean velocity at each section is normalized with the reference velocity (U_0) of 15 m/s. The flow at entrance, at a location of $X/D_v = 0.03$, has a uniform mean velocity profile. The second profile, at $X/D_v = 0.47$, lies within the first recirculation region upstream of the valve. It may be noted that the values become negative near the cylinder wall indicating reverse flow there. The maximum value in the profiles is located close to the valve tip. Figure 5.6-1(2) also shows a rapid deceleration of fluid near the valve stem, accompanied by a corresponding acceleration in the valve lift region (near valve tip). The profiles downstream, at $X/D_v = 0.63$, 0.81 and 2.21, show the flow characteristics downstream of the valve, indicating the presence of a second recirculation there. The last one, at $X/D_v = 9.6$, shows the velocity distribution further downstream of the valve and near the exit boundary. It is almost uniform, indicating the recovery of the flow.

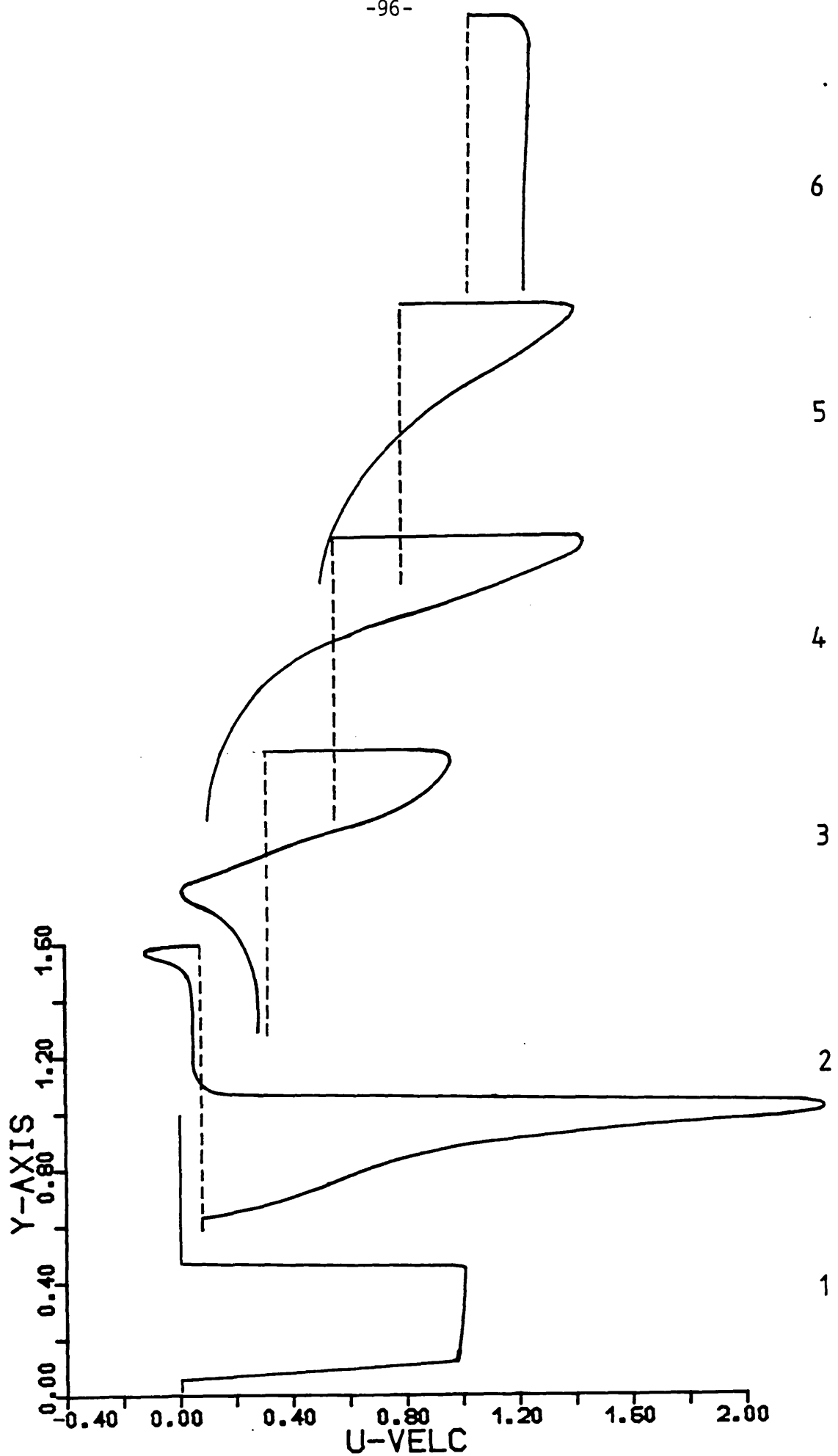


Figure 5.6-1 Radial profile of axial velocity at six streamwise locations for A; $L_V D_V = 0.15$.

The development downstream of the valve can be conveniently summarised by an inspection of the profile of centreline axial velocity and the streamline pattern. Figure 5.6-2 presents the centreline axial velocity for $L_v/D_v = 0.15$ and indicates the region of reverse flow in the lee of the valve, which extends to $X/D_v = 2.548$, and the subsequent development as the wake profile reforms towards that of a pipe flow.

In Figure 5.6-3 is shown the centreline profiles of axial velocity for L_v/D_v of 0.1, 0.15 and 0.3.

Figure 5.6-4 shows the streamline pattern corresponding to the profiles of Figure 5.6-1. The plots have been constructed by evaluating the stream function:

$$\psi(x,y) = \int_0^r u(x,y) y \, dy \quad .(5.6-1)$$

Axial variation of wall-pressure is displayed in Figure 5.6-5. It is

given by plotting the non-dimensionalised pressure,

$$\frac{P - P_0}{0.5\rho U_0^2} \text{ versus } X/D_0, \text{ where } P_0 \text{ is the wall pressure at the inlet}$$

duct and U_0 is the inlet mean velocity.

The flow outside the separation streamline must accelerate as it passes over the maximum width of the recirculation. The fall in static pressure which accompanies this acceleration is clearly depicted in Figure 5.6-5.

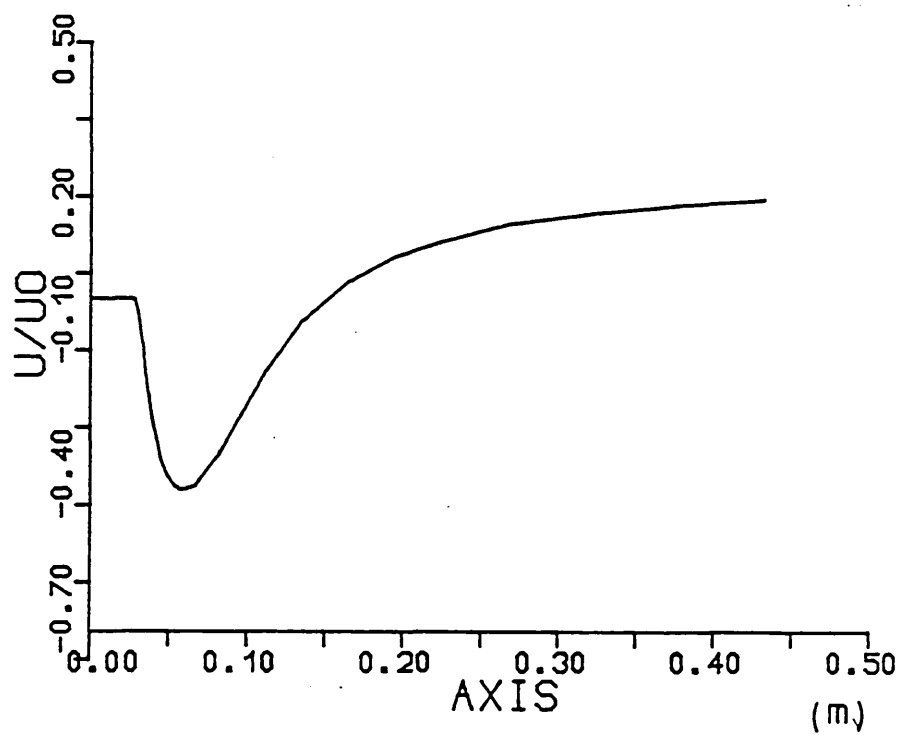


Figure 5.6-2 Centreline profile of axial velocity for case A

$$L_V/D_V=0.15 .$$

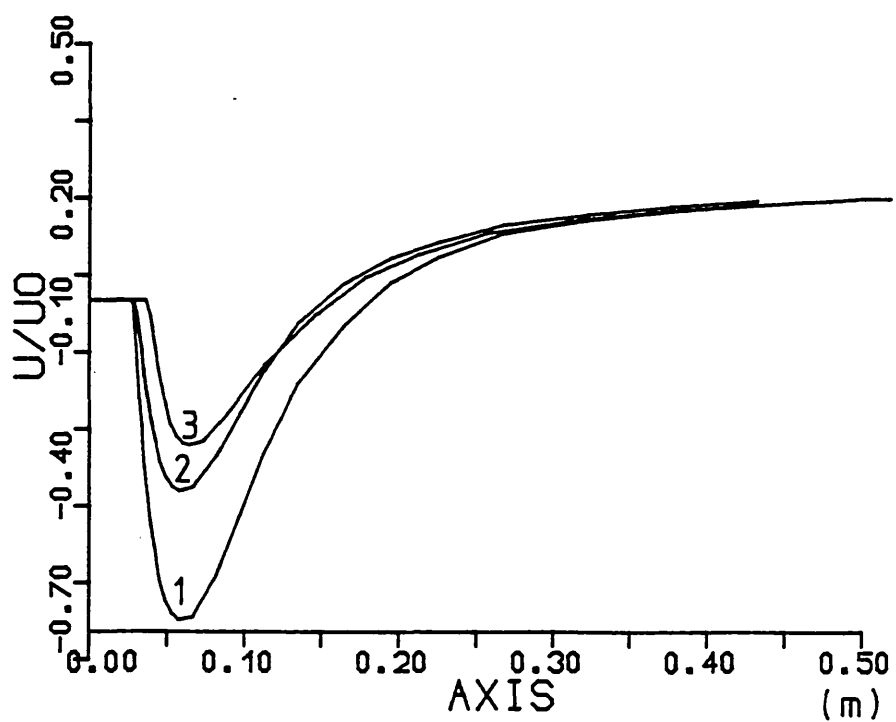


Figure 5.6-3 Centreline profiles of axial velocity for flow

through a seatless valve with three valve locations

(1) $L_V/D_V=0.1$, (2) $L_V/D_V=0.15$, (3) $L_V/D_V=0.3$.

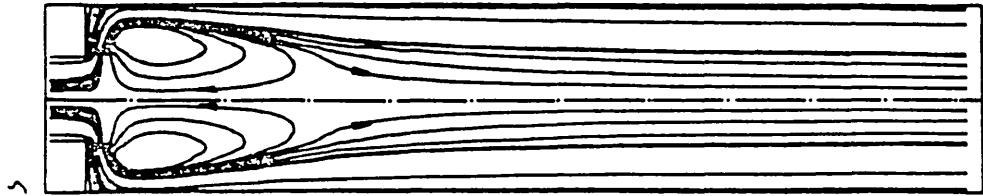


Figure 5.6-4 Streamlines of flow through a seatless valve

$$L_v/D_v = 0.15 .$$

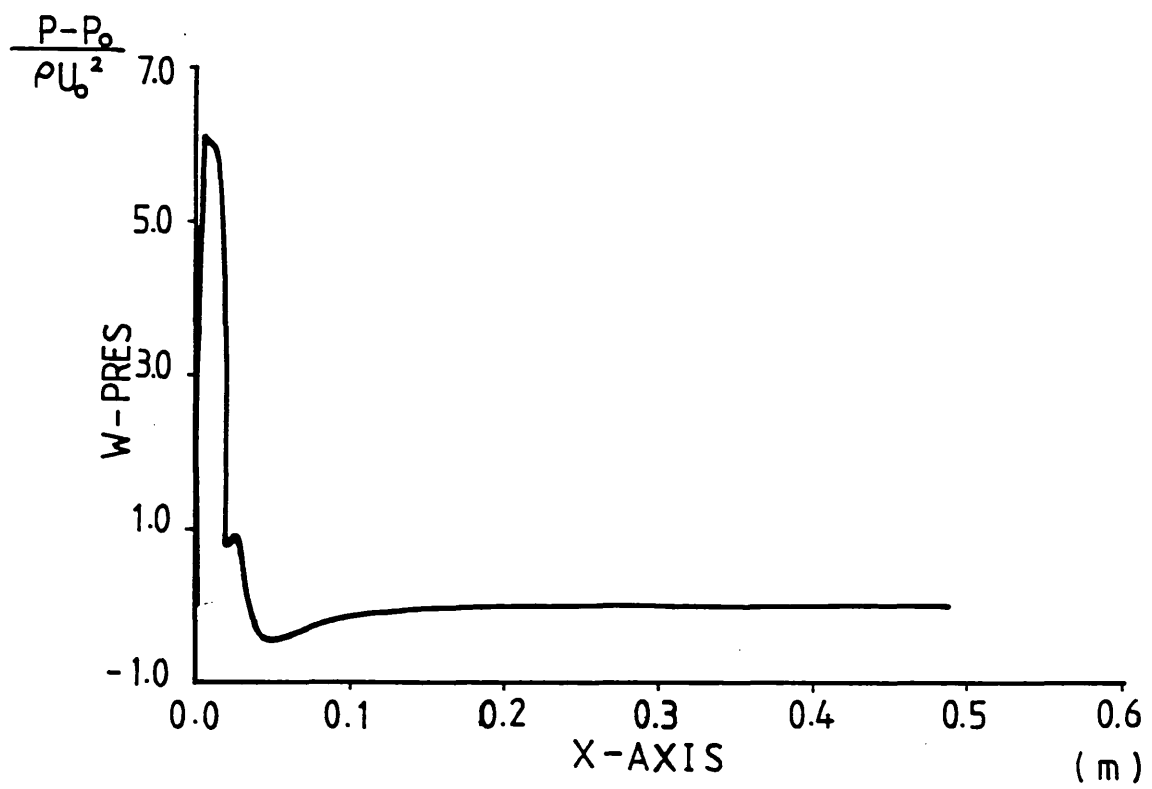


Figure 5.6-5 Axial variation of wall-pressure.

Turbulence Energy

The centreline profiles of turbulence intensity and turbulence kinetic energy normalized by U_0 and ρU_0^2 respectively are shown in Figure 5.6-6 and 5.6-7.

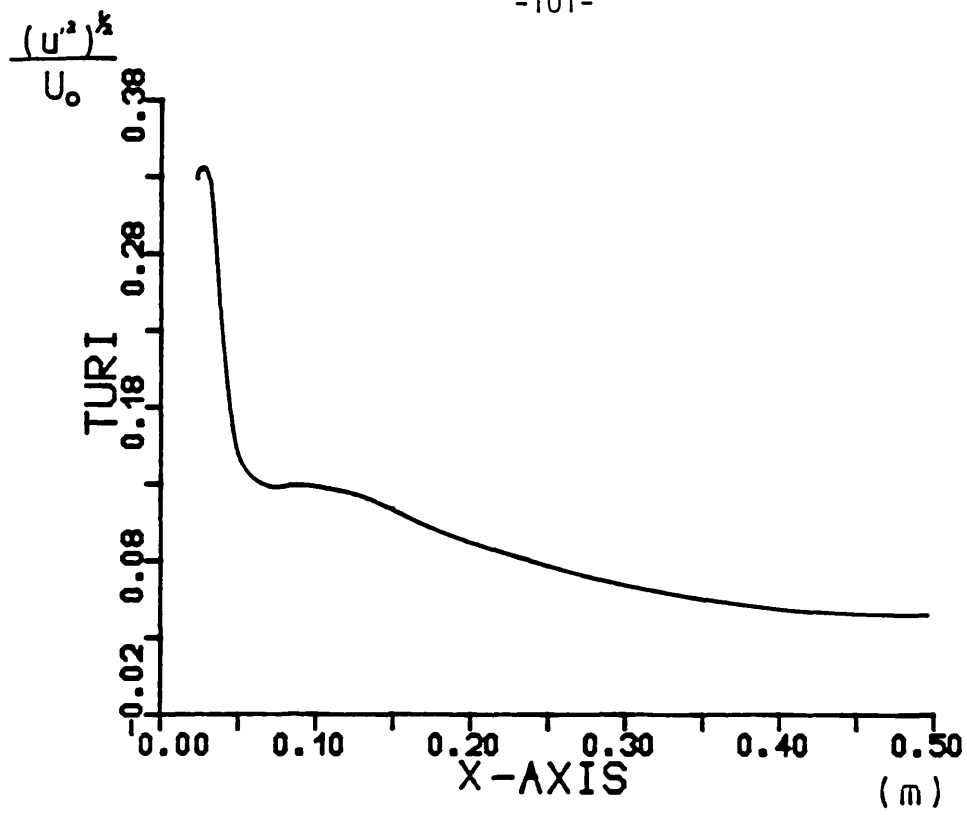


Figure 5.6-6 Centreline values of turbulence intensity.

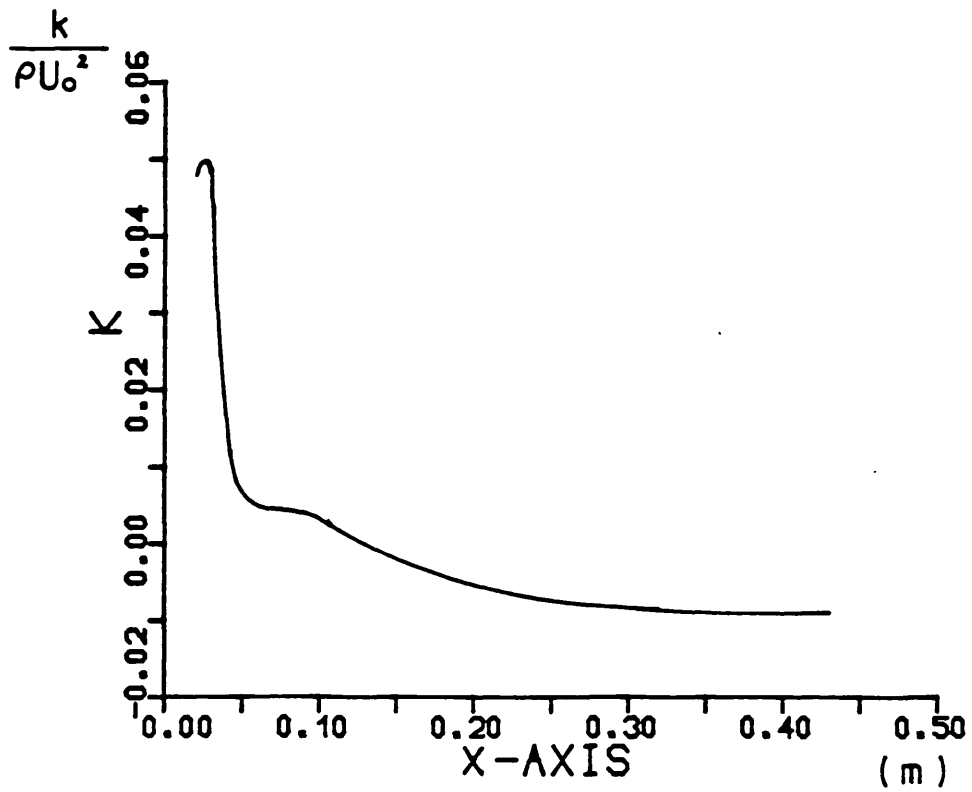


Figure 5.6-7 Centreline values of turbulent kinetic energy.

Discharge Coefficient

The variation of the computed discharge coefficient with valve lift at a constant mass flow rate of 9.5×10^{-2} kg/s for the flow through a valve entering an enlarged cylinder (case A), and that entering the atmosphere (case B) is shown in Figure 5.6-9. They are compared with the experimental data of Annand, which has only covered the latter situation.

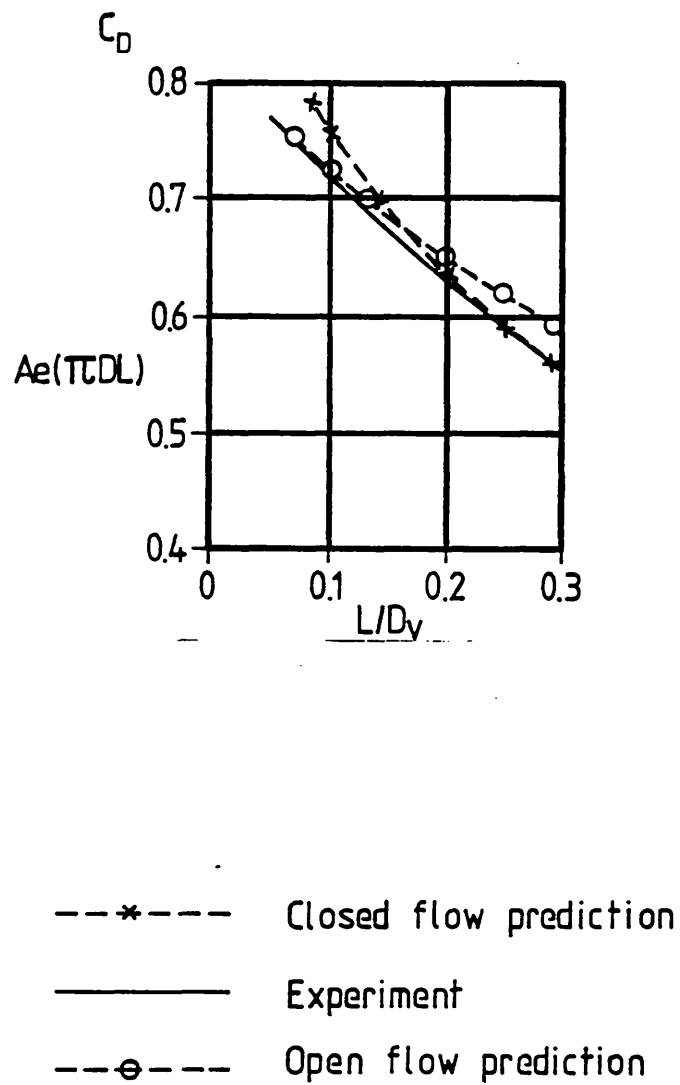


Figure 5.6-8 Comparison of discharge coefficient of closed and open flow situations.

5.7 Discussion

The aspects of the flowfield that are of interest in this study are the velocity profile, wall-pressure, the level of turbulence, the discharge coefficient and the influence of valve lift on them.

Mean Velocity and Wall Pressure

The velocity distribution represented in Figure 5.6-1 displays that the flow near and downstream of the valve qualitatively resembles those observed by Bicen (1983) in a two- or four-stroke engine with the jet-like flow driving two vortices, one in the corner between the wall and the cylinder head, and the other downstream of the valve near the cylinder axis. This is also confirmed in Figure 5.6-4.

The centreline profile of axial velocity in Figure 5.6-2 also shows the same general characteristics. There is a region of recirculating flow where the mean velocity is negative, one of acceleration and another one further downstream of nearly uniform or slowly decreasing velocity. The influence of the valve lift on the centreline velocity shown in Figure 5.6-3 reveals that the maximum negative value decreases with increasing valve lift.

The axial variation of wall-pressure in Figure 5.6-5 displays the minimum pressure which exists at the section of the 'eye' of the recirculation zone. The flow outside the separation streamline must accelerate as it passes over the maximum width of the recirculation zone downstream of the valve. The fall in static pressure which accompanies this acceleration can also be seen from this diagram; later the pressure

recovers.

Turbulence Energy

The centreline profile of turbulence kinetic energy and turbulence intensity depicted in Figures 5.6-6 and 5.6-7 show that large $k/\rho U_0^2$ and $(u^2)^{1/2}/U_0$ values are located behind the valve, because the recirculation there generates intense turbulence.

Discharge Coefficient

The variation of the predicted values of the discharge coefficient with the valve lift for cases A and B is shown in Figure 5.6-9. The experimental data for case B, which were measured by Annand, are also displayed in the same figure. They show a similar trend of variation.

The calculation was carried up to a dimensionless lift (L_v/D_v) of 0.3, a range which is commonly used in practice, and the Reynold number based on the inlet diameter D_v is 1.0×10^5 . High discharge coefficients are obtained at low lifts, suggesting that the flow boundaries remain attached to the walls of the valve passage and fill the entire exit plane; and that any separation occurring in the passage is small at a low lift, while at high lifts the flow separation near the tip of the valve head becomes more pronounced which lowers the discharge coefficients. On comparison of the predictions with the experimental data, the agreement is satisfactory when L_v/D_v is above 0.15 for the closed flow situation (Figure 5.6-9). In the open flow situation, the agreement seems to be fair up to 1.5 of L_v/D_v , but then the predictions start to deviate. The maximum discrepancy is about 5% for the open flow, 2.7% for the closed

flow.

The comparison of predicted discharge coefficient for the two cases is shown in Figure 5.6-9, revealing that the difference between the discharge coefficient of flow through a valve entering an enlarged cylinder and flow entering the atmosphere is not large, indicating that the data available in flow entering open atmosphere can be used with confidence in determining the discharge characteristics of inlet ports and valves.

5.8 Closure

In this chapter, flow of air through a seatless poppet valve has been simulated. The predictions of discharge coefficient have been compared with the experimental data of Annand and the agreements have been found to be generally fair. It has also revealed that the flow discharge coefficient at an open flow situation is close to the closed one; an experiment for open flow situation can then be used to study the flow characteristics.

The predictions have been carried beyond the experimental information. Field values of some flow variables have been provided. It resembles those observed by Bicen (1983) in two and four-stroke engines with the jet-like flow. Steady flow testing may, therefore, be used for understanding the fundamentals of the valve flow.

CHAPTER 6 - FLOW OF AIR THROUGH A VALVE WITH AN ANGLED SEAT

6.1 Introduction

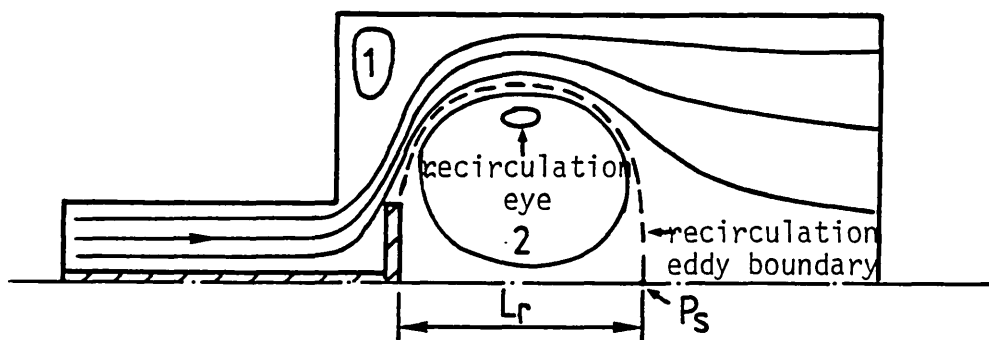
In the previous chapter, the numerical simulation of flows through a seatless poppet valve was presented. The present chapter describes an extension for flows through a valve with an angled seat.

The seatless valve, which has a simpler geometry than that of a valve with an angled seat, has served as a preliminary study to obtain some insight into the characteristics of a valve flow. In the present chapter, the investigation described represents an attempt to examine the effect of valve-seat width, seat angle and valve lift on the flow pattern and discharge coefficient.

The description of the phenomena to be predicted is given in section 6.2. In section 6.3 are given the geometry, dimensions and the cases considered. The detailed boundary conditions and the grid refinement are described in section 6.4 and 6.5. The predicted results are presented in section 6.6. Some of the results are compared with the experimental data of Annand, followed by discussion in section 6.7. The chapter concludes by summarising the main findings in section 6.8.

6.2 Phenomena to be Predicted

In the present study, consideration is given to a steady turbulent flow from an inlet duct through a valve with an angled seat, into an enlarged cylinder. Separated recirculating flows are established upstream and downstream of the valve (Figure 6.2-1); there are two vortices, one in the corner between the cylinder wall and the step; and the other in the lee of the valve near the axis. The influence of the valve geometry on the discharge coefficient, and on the flow pattern is studied.



- 1 recirculation zone behind the step
- 2 recirculation zone downstream of the valve
- L_r recirculation zone length
- P_s rear stagnation point

Figure 6.2-1 Mean streamline pattern for a seatless valve.

6.3 Geometry, Dimensions and Cases Considered

The geometry of the inlet port and conical-valve arrangement is depicted schematically in Figures 6.3-1 and 6.3-2. The dimensions of the model valve are the same as those of the apparatus used in the experiments of Annand; they are:

head diameter (D_v)	0.045m.
stem diameter (d)	0.00095m.
seat width (w)	0.0016m.
valve thickness	0.0016m.

Dimensions of the inlet port and valve geometry:

$$D_o = 0.045m, \quad D = 0.1m,$$

$$D_{vs} = D_o + 2w = 0.0482m, \quad L = 0.5m,$$

$$L_v = 11ft, \quad w = \text{seat width},$$

$$\theta = \text{seat angle}.$$

The study concerns the variations of three geometric parameters: they are the ratio of valve lift to valve diameter, L_v/D_v , (case 2-8); that of valve seat width to valve diameter, w/D_v , (cases 9-15); and the valve seat angle, θ , (cases 16-21) as summarized in Table 6.3-1. (case 1 is performed as a basic case).

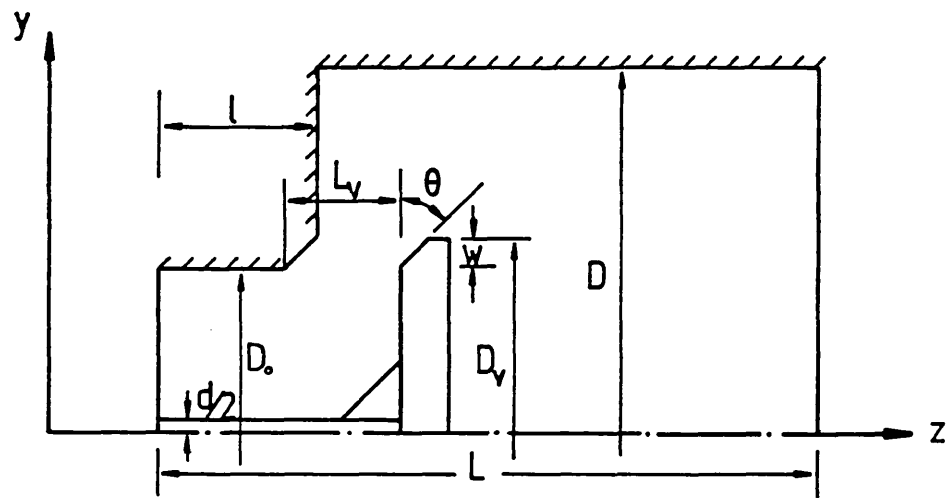


Figure 6.3-1 The geometry of an inlet valve.

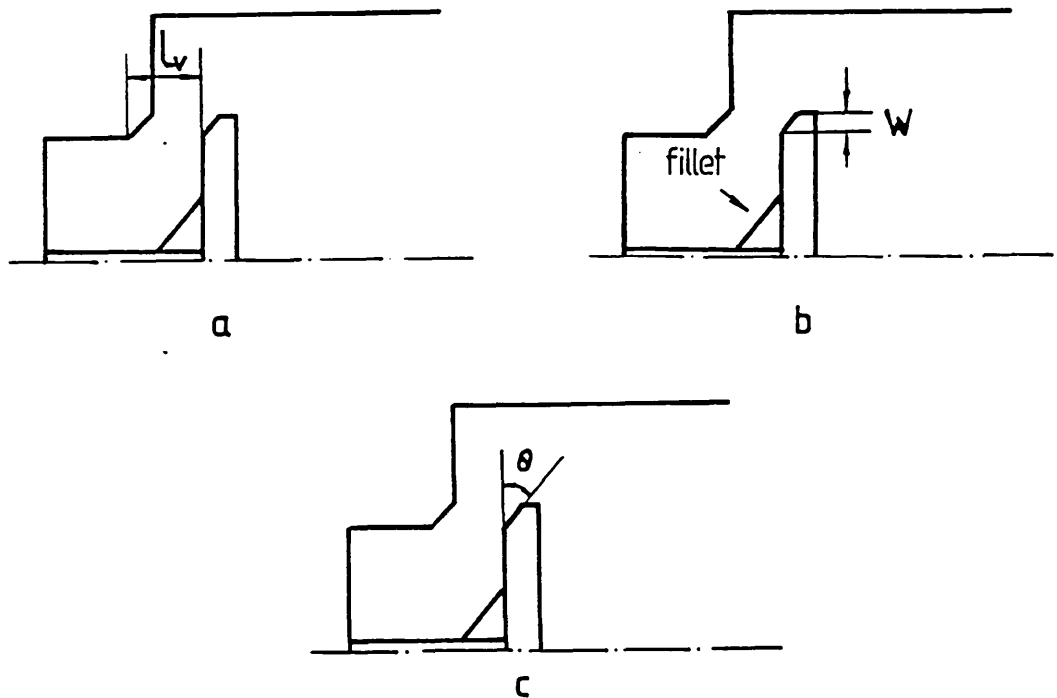


Figure 6.3-2 The details of the valve parameter

L_v : lift, w : seat width, θ : seat angle.

Table 6.3-1 Characteristics of the 21 cases studied.

Case	Θ	w/D_v	L_v/D_v
1	45°	0.035	0.16
2	45	0.035	0.053
3	45	0.035	0.071
4	45	0.035	0.106
5	45	0.035	0.16
6	45	0.035	0.213
7	45	0.035	0.248
8	45	0.035	0.284
9	45	0.068	0.053
10	45	0.068	0.071
11	45	0.068	0.106
12	45	0.068	0.16
13	45	0.068	0.213
14	45	0.068	0.248
15	45	0.068	0.284
16	30	0.035	0.16
17	35	0.035	0.16
18	40	0.035	0.16
19	45	0.035	0.16
20	50	0.035	0.16
21	60	0.035	0.16

6.4 Boundary Conditions

In Figure 6.4-1 are shown the boundaries of the computational domain; all conditions on them are described below.

Axis of Symmetry

Along the symmetry axis, the normal velocity, v , and all normal gradients of other variables, are zero.

Inlet Plane

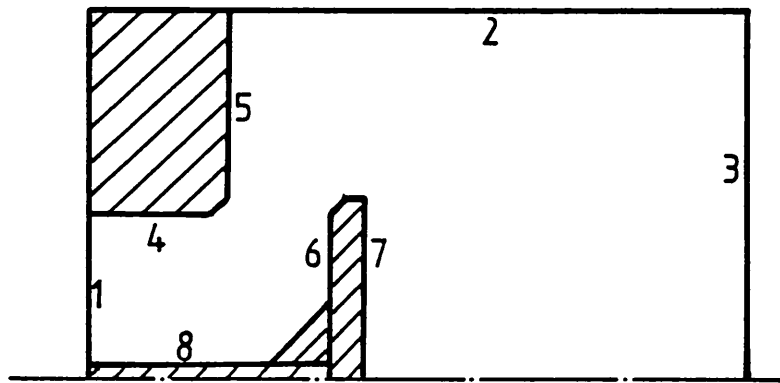
At the inlet section, a uniform velocity profile at 15 m/s across the duct is prescribed, the turbulence-energy, k , is $0.675 \text{ m}^2/\text{s}$ ($0.003 \times W_{In}^2$), and its dissipation rate is $1.664 \text{ m}^2/\text{s}^3$ ($0.09 \times k_{In}^{1.5} / (0.003 \times D/2)$).

Outlet Plane

A constant and uniform pressure condition, $P = 0 \text{ kg/m}^2$, is imposed.

Walls

The regions near all walls, ie. cylinder wall, duct wall, and valve-face and -stem, are treated by use of the wall function described in section 2.5.



- | | |
|-----------------|----------------------|
| 1 inlet plane | 5 cylinder head wall |
| 2 cylinder wall | 6 valve face |
| 3 outlet plane | 7 " " |
| 4 duct wall | 8 valve stem |

Figure 6.4-1 Boundary region distribution

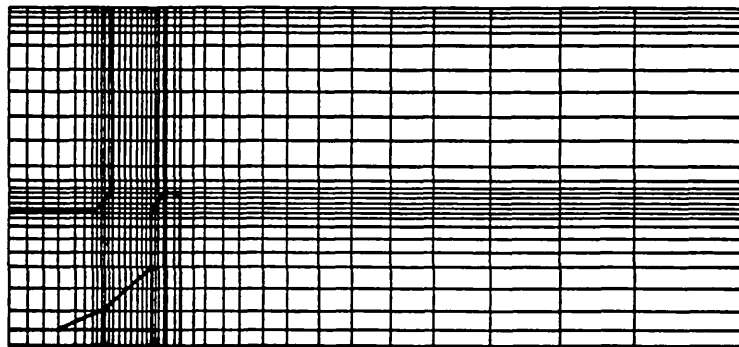
6.5 Grid Refinement

Since the present problem has a complicated geometry, the grid lines are distributed non-uniformly over the flow domain. A grid distribution for case 1, as shown in Figure 6.5-1, is provided here in order to illustrate its non-uniformity. It can be seen that closer spacings are used in the more important regions, i.e. valve lift, near the conical seat of the valve and the cylinder wall.

Grid-refinement studies have been performed for flow through a seatless valve in chapter 5; the grid employed in chapter 5 is taken as a minimum grid number for the grid-dependence study. This was carried out, mainly for determining the influence of the grid number in the conical seat and lift regions on the velocity profiles, and the influence of the grid number on the discharge coefficient. Table 6.5-1 displays the predicted velocity at section Z for four selected grids in the conical-seat region of case 1. It can be seen by reading the line horizontally that the velocities at the same position behind valve do not change significantly beyond 26 x 51 grids (26 y-nodes with 4 nodes in the conical-seat region and 51 z-nodes). Table 6.5-2 shows the influence of the grid-nodes in the lift region on the velocity profile at section Z, downstream of the valve. Three kinds of grid are investigated. It is seen that the velocity values do not change significantly beyond 26 x 51 nodes (26 y-nodes and 51 z-nodes with 7 grid nodes in the lift region).



a



↑ refer to $Z = 0.03305$

b

Figure 6.5-1 Grid distribution of the calculation domain:

(a) for the whole domain;

(b) part of the domain around the valve.

Table 6.5-1

Total grid No.	23x40	26x44	26x51	30x52
Grid No. in seat region	2	4	5	8
Velocity W/W_{in}	2.010E-1	4.441E-1	4.451E-1	4.459E-1
(at $Z=0.03305$)	3.011E-1	4.515E-1	4.521E-1	4.527E-1
Y ↑	3.901E-1	4.567E-1	4.574E-1	4.577E-1
	4.215E-1	4.647E-1	4.655E-1	4.659E-1
	-1.713E-1	-1.924E-1	-1.934E-1	-1.937E-1
	-1.117E-1	-8.735E-2	-8.747E-2	-8.751E-1
	-6.518E-2	-5.645E-2	-5.651E-2	-5.687E-2
	-3.714E-2	-3.002E-2	-3.015E-2	-3.019E-2
	-1.715E-2	-1.395E-2	-1.407E-2	-1.415E-2

Table 6.5-2

Total grid no.	26x42	26x51	26x55
Grid No. in lift region	5	7	8
Velocity W/W_{in}	4.431E-1	4.441E-1	4.449E-1
(at $z=0.03305$)	4.503E-1	4.515E-1	4.521E-1
Y ↑	4.521E-1	4.533E-1	4.540E-1
	4.642E-1	4.651E-1	4.657E-1
	3.539E-1	3.553E-1	3.561E-1
	-1.911E-1	-1.924E-2	-1.931E-1
	-1.489E-1	-1.511E-1	-1.525E-1
	-1.190E-1	-1.210E-1	-1.217E-1
	-8.733E-2	-8.775E-2	-8.782E-2
	-6.437E-2	-6.551E-2	-6.560E-2
	-4.679E-2	-4.711E-2	-4.719E-2
	-3.007E-2	-3.011E-2	-3.017E-2
	-1.789E-2	-1.903E-2	-1.913E-2
	-1.381E-2	-1.397E-2	-1.399E-2
	-8.497E-10	8.503E-10	-8.511E-10

Figure 6.5-2 represents the influence of the grid number on the discharge coefficient, the values of which are shown in Table 6.5-3. It can be seen that the discharge coefficient does not change significantly beyond 26 x 51 grid-nodes. The final grid employed is 26x51 and by the above tests is regarded as sufficiently grid-independent. A total of 26 y-grids are used to span 0.045m, of which 4 nodes are used for 0.0016m conical-seat, as shown. 51 Z-nodes are used to span 0.6m, of which 4 grid-nodes are used for the 0.0016m valve seat, 7 nodes are used in the lift region.

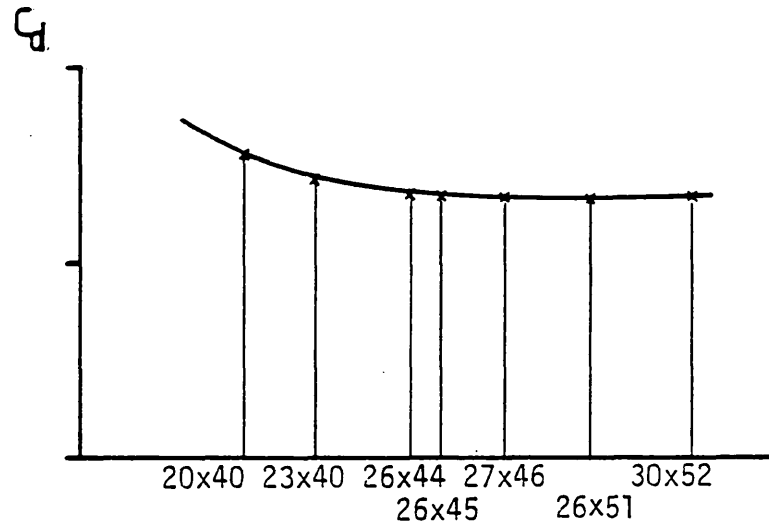


Figure 6.5-2 The influence of grid number on the discharge coefficient.

Table 6.5-3 The influence of grid-nodes on the discharge coefficient.

$L_V/D_V = 0.16$, $W/D_V = 0.035$, $\theta = 45^\circ$.

Grid No.	20x40	23x40	26x44	26x45	26x51	30x52
Discharge coefficient C_D	0.7831	0.7173	0.6729	0.6664	0.6645	0.6545

6.6 Presentation of the Main Results

6.61 Radial Distribution Axial Velocity

Since many cases involving many dependent and auxillary variables have been studied. It is impossible to show all results: only a limited selection of velocity profiles, as shown in Figure 6.6-1, are displayed and discussed.

Figure 6.6-2 represents the radial profiles of axial velocity at six axial stations for the dimensionless valve lift, $L_v D_v$, of 0.106 (i.e. case 4); Figures 6.6-3 and 6.6-4 show the velocities at the same locations for valve lifts of 0.16 and 0.284 (i.e. cases 5 and 8). The velocity at each section is normalized by a reference velocity (U_0).

From these three diagrams, it is seen that the velocity profiles at 1 are nearly uniform at entrance. The second profiles at 2, $Z/D_v = 0.52$ respectively in Figures 6.6-2, -3 and -4, show the recirculating characteristics upstream of the valve. The negative values appear near the cylinder wall indicating reverse flow. It is seen that the smaller the valve lift the larger the recirculation zone, and the larger the maximum axial velocity. Radial profiles at locations 3, 4 and 5 ($(Z-Z_v)/D_v = 0.55, 0.876$ and 2.186 , shown in these three Figures) lie within the second recirculation zone downstream of the valve. The last radial profiles at location 6, $(Z-Z_v)/D_v = 7.846$, presents the velocity distributions further downstream. The profiles are approximately uniform, indicating the recovery of the flow.

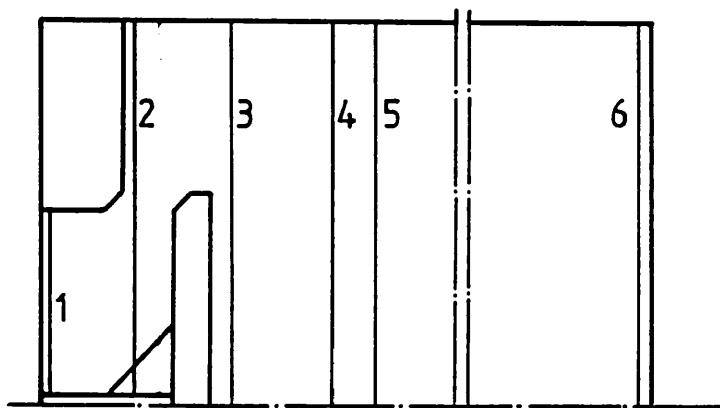


Figure 6.6-1 The chosen location for the velocity sketch.

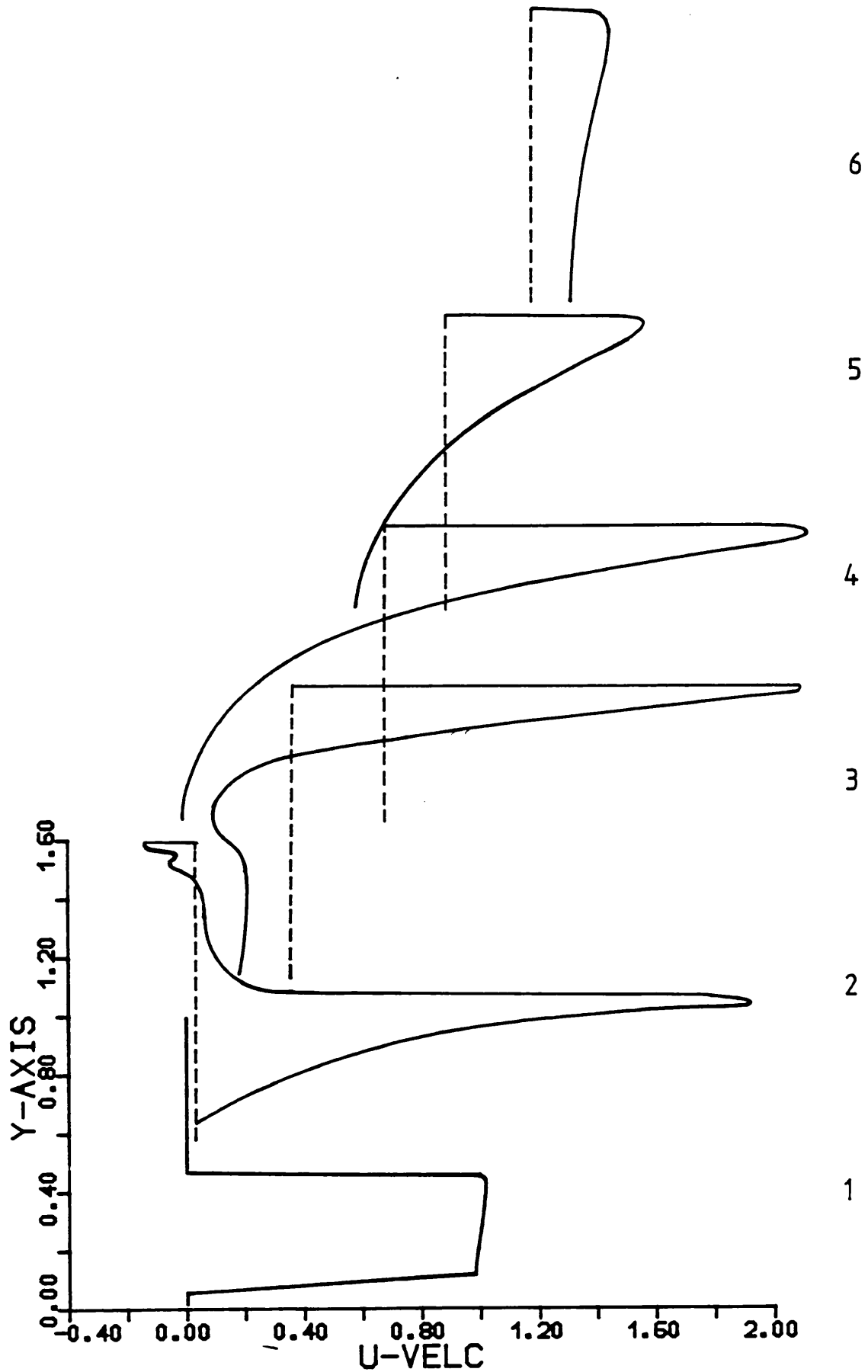


Figure 6.6-2 Radial profiles of axial velocity of locations 1, 2, 3, 4, 5, 6 for valve lift $L_v/D_v = 0.106$, $\theta = 45^\circ$, $W/D_v = 0.035$.

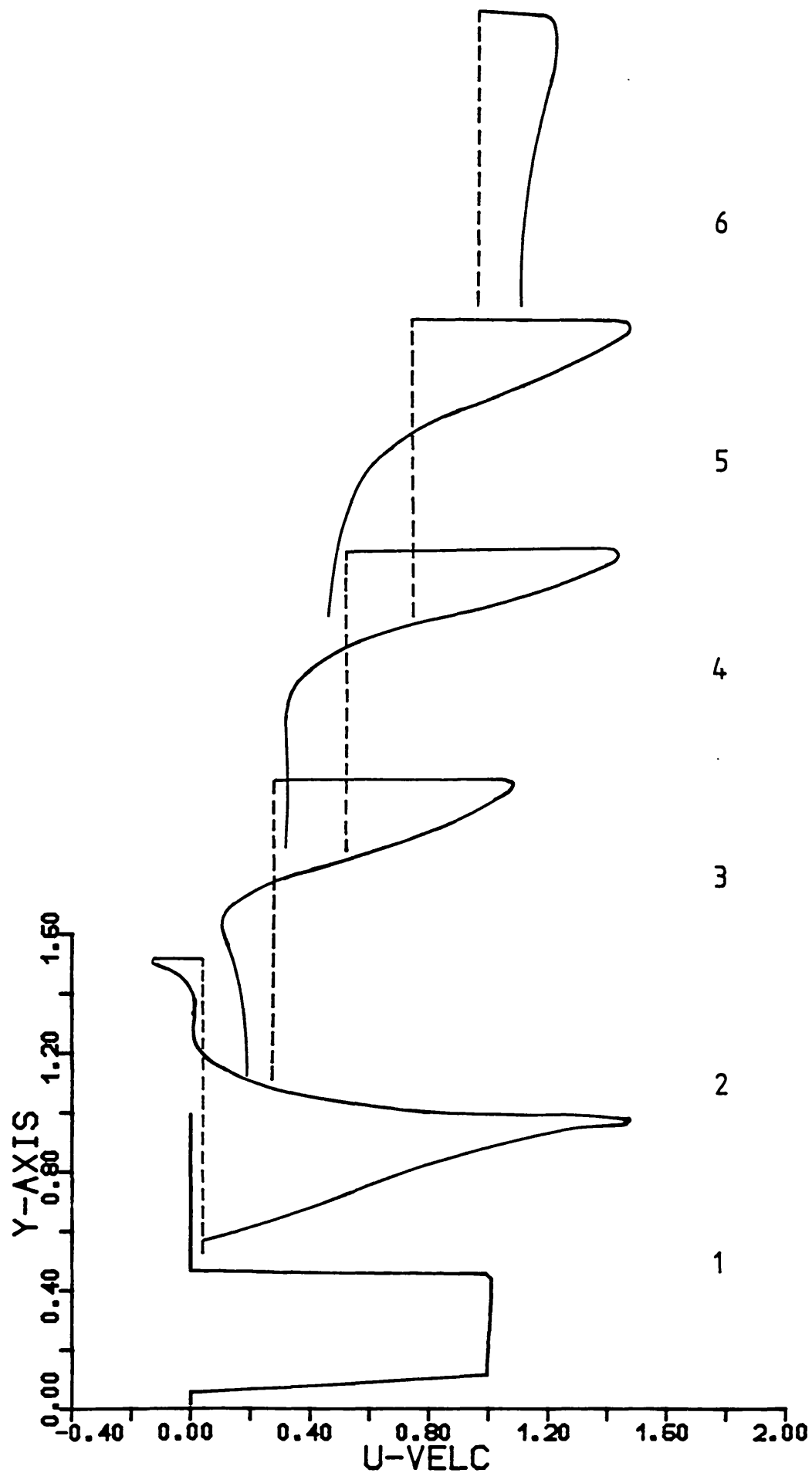


Figure 6.6-3 Radial profiles of axial velocity at six locations for valve lift $L_v/D_v \approx 0.16$, $\theta \approx 45^\circ$, $w/D_v \approx 0.035$.

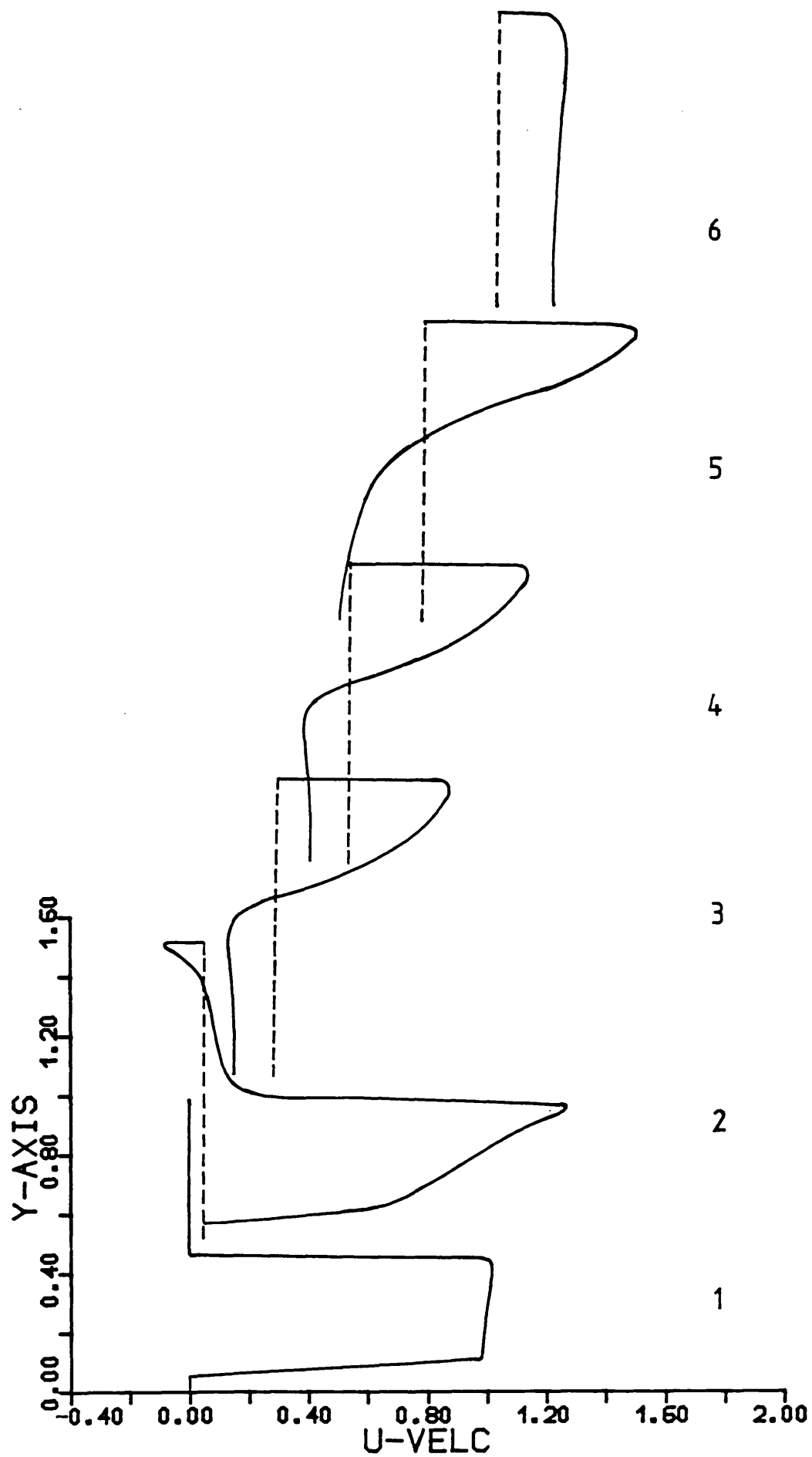


Figure 6.6-4 Radial profiles of axial velocity at six locations for valve lift $L_v/D_v = 0.284$, $\theta = 45^\circ$, $w/D_v = 0.035$.

6.62 Radial Velocity

In Figure 6.6-5, the distributions of radial velocity component v_r near the valve exit, are compared for three different lifts. It can be seen that a higher v_r -velocity is associated with a smaller lift.

6.63 Centreline Distribution of Axial Velocity

In Figure 6.6-6 is shown the centreline distribution of axial velocity component for the same three (lift), cases. It shows that the smaller the valve lift, the larger the maximum negative axial velocity.

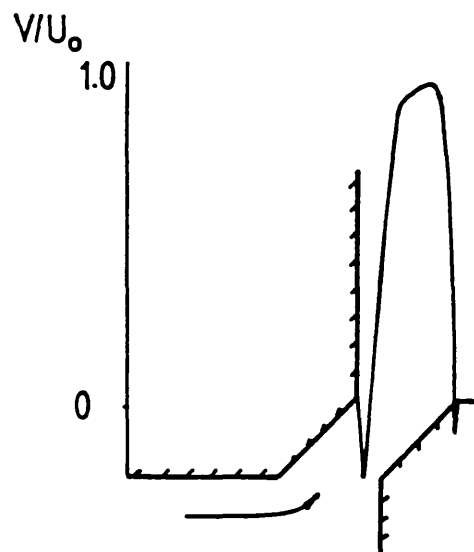
6.64 Streamline Pattern and Recirculating Mass Flow Rate

Figure 6.6-7 shows the streamline pattern for case 1 ($L_v/D_v = 0.16$). They were calculated by:

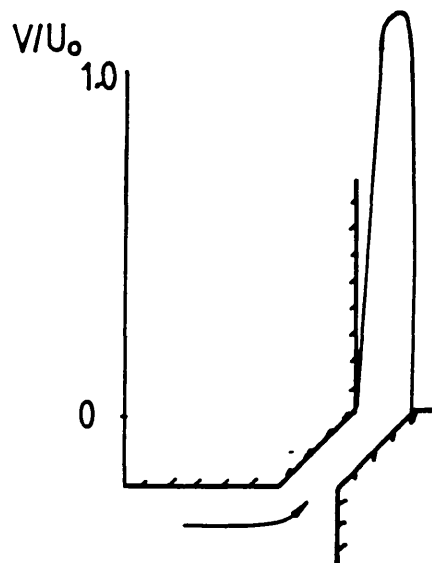
$$\dot{m} = 2\pi\rho \int_0^r U(Z,r) r dr \quad , (6.6-1)$$

The spatial distribution of the streamlines enables the size and the shape of the recirculation zone to be evaluated. Figure 6.2-1 In section 6.2 has already illustrated the recirculation zone, which may be characterised by a boundary streamline which represents $\dot{m} = 0$.

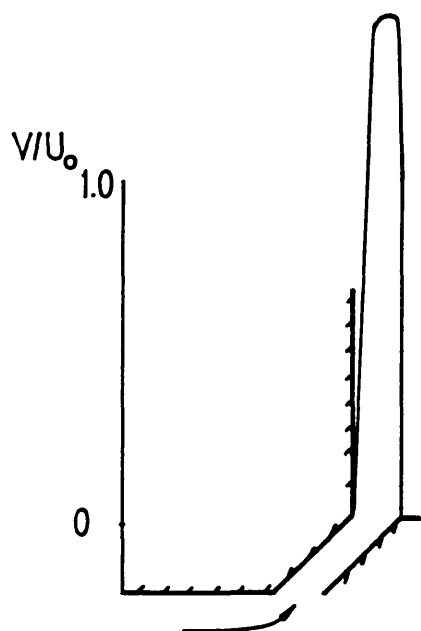
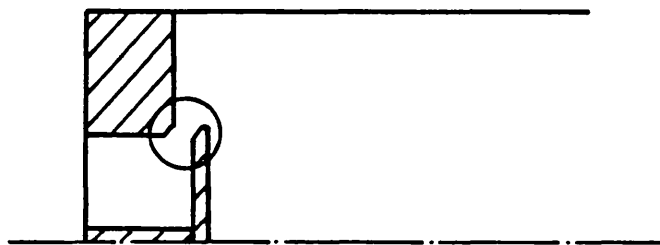
Consideration of the mass flow rates trapped within the recirculation zone will provide a measure of the relative strength of the vortex motion; and its magnitude, $\dot{m}_r$ (recirculation mass flow rate), may be obtained from the integration:



$$L_V/D_V = 0.284$$



$$L_V/D_V = 0.16$$



$$L_V/D_V = 0.106$$

Figure 6.6-5 Distribution of radial velocity at exit from valve for three lifts: $L_V/D_V = 0.284$, 0.16 , and 0.106 .

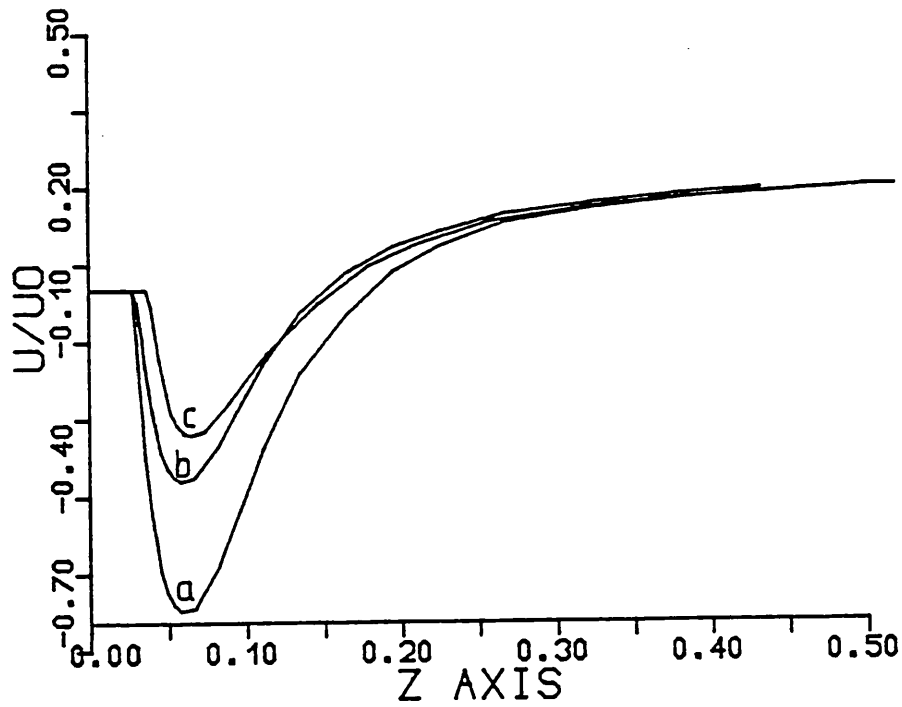


Figure 6.6-6 Centreline profile of axial velocity for flow through a valve with 45 deg angled seat with three valve locations:
(a) $L_V/D_V = 0.106$, (b) $L_V/D_V = 0.16$, (c) $L_V/D_V = 0.284$.



Figure 6.6-7 Streamlines of flow through a seated valve
 $L_V/D_V = 0.16$, $w/D_V = 0.035$, $\theta = 45^\circ$.

$$\dot{m}_r = 2\pi\rho \int_0^{r_l} U(Z,r) r dr \quad .(6.6-2)$$

where r_l is the radial position of the zero velocity boundary. $\dot{m}_r$ is the maximum negative mass flow rate.

Figure 6.6-8 shows the axial distribution of the normalized recirculating mass flow rate, $\dot{m}_r/\dot{m}_{in}$. $\dot{m}_{in}$ is the mass flow rate at entry duct.

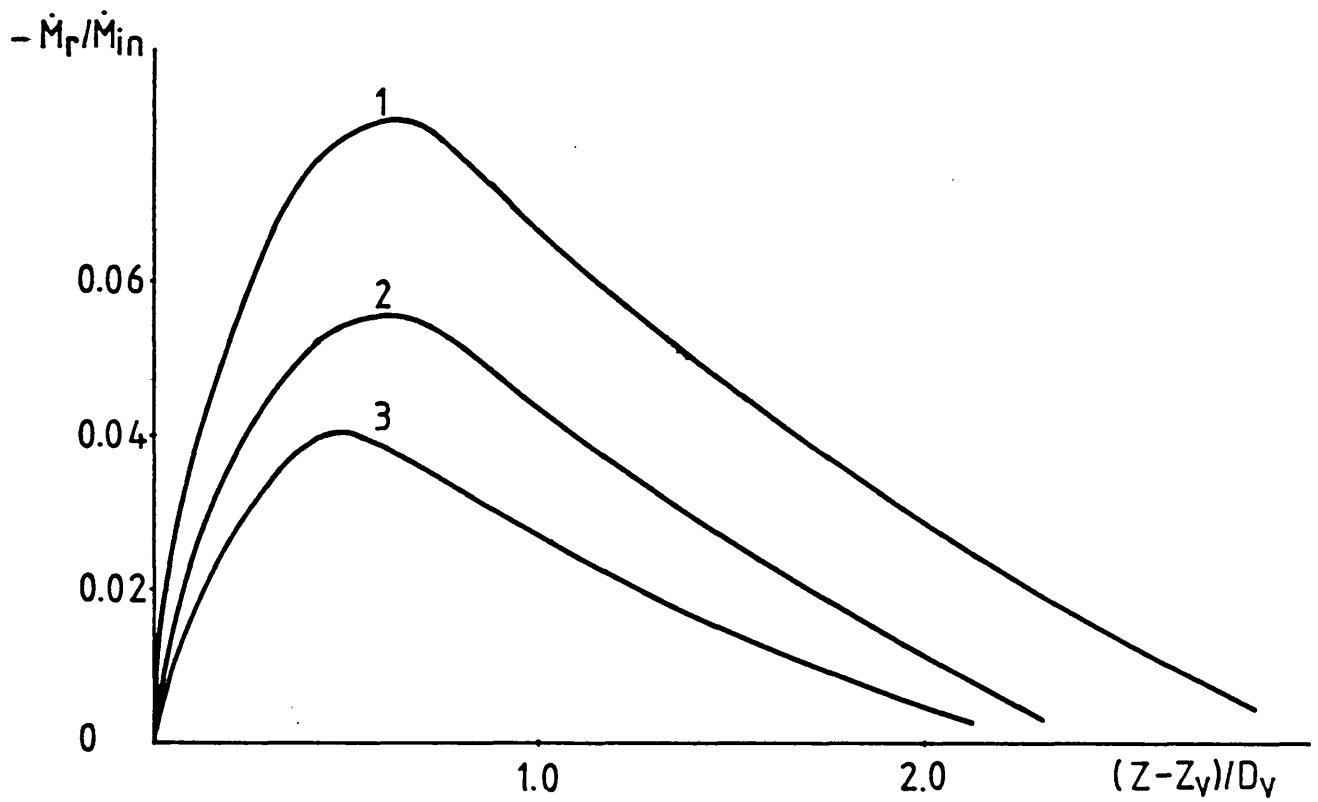
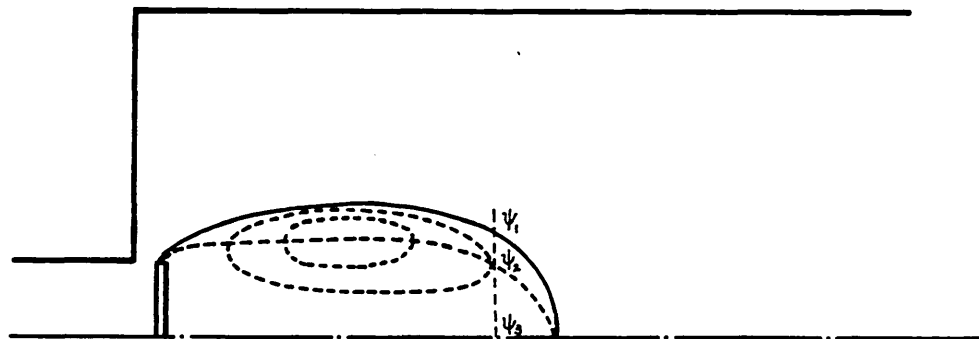


Figure 6.6-8(a) Variation of the recirculation flow rate with axial distance for three lifts: (1) $L_V/D_V = 0.106$, (2) $L_V/D_V = 0.16$, (3) $L_V/D_V = 0.284$.



$$\dot{m} = |\psi_1 - \psi_2| = |\psi_2 - \psi_3|$$

$$\psi_2 = \psi_2(z)$$

Figure 6.6-8(b) Streamline pattern in the recirculation zone.

Table 6.6-1 compares the maximum recirculating mass flow rate, $\dot{m}_{r,max}/m_{In}$, and the length, L_r , of the reverse flow zone downstream of the valve for three valve lifts. It shows that $\dot{m}_{r,max}/m_{In}$ for $L_v/D_v = 0.284$ is only 51.7% of that for $L_v/D_v = 0.106$.

Table 6.6-1 The maximum recirculating mass flow rate and the length of the reverse flow zone for three valve lifts

case	dimensionless lift L_v/D_v	$\dot{m}_{r,max}/m_{In}$	$L_r(m)$
4	0.106	-0.5969	0.1368
5	0.16	-0.4126	0.1153
8	0.284	-0.3088	0.1097

6.65 Pressure

The calculated isobars for flow through a valve with an angled seat ($L_v/D_v = 0.16$) are shown in Figure 6.6-9(a). The minimum pressure appears at the eye of the recirculation zone.

In Figure 6.6-9(b) are shown the axial variations of wall-pressure for cases 4, 5 and 8 for three dimensionless lifts of 0.106, 0.16 and 0.284. It can be seen that as the valve lift is reduced the minimum wall-pressure reduces accordingly. The pressure recovery in the downstream region can also be observed.

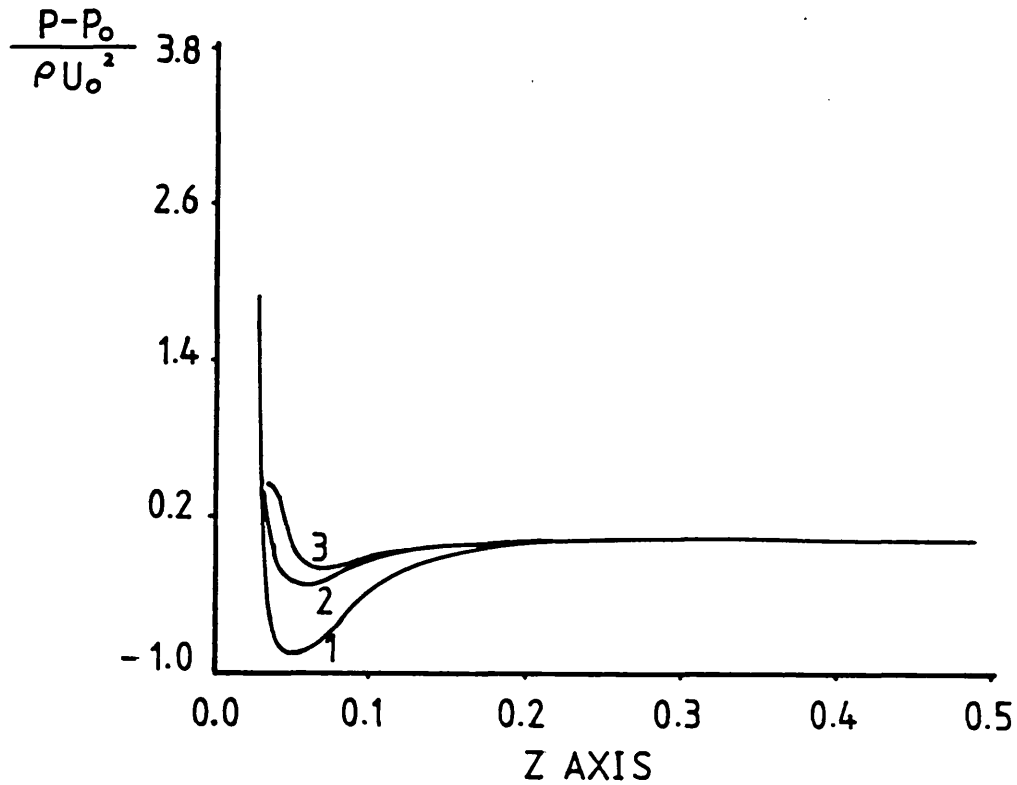


Figure 6.6-9(b) The axial variation of the wall-pressure downstream of the valve for three lifts: (1) $L_V/D_V = 0.106$, (2) $L_V/D_V = 0.16$, (3) $L_V/D_V = 0.284$.

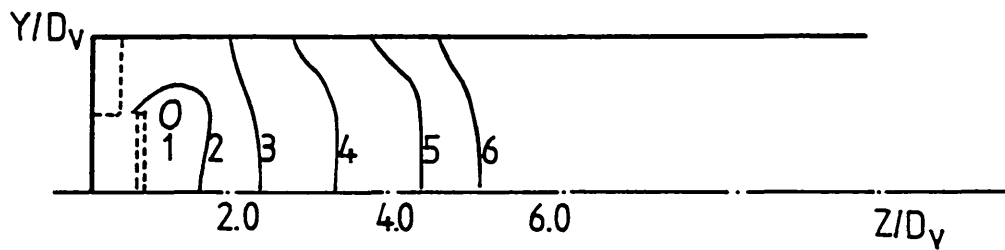


Figure 6.6-9(a) Calculated isobars for flow through a valve with an angled seat $\theta = 45^\circ$ $w/D_V = 0.053$ $L_V/D_V = 0.16$.
 1 -100 ; 2 -80 ; 3 -60 ; 4 -40 ;
 5 -20 ; 6 -10 (percentage of $\frac{1}{2}\rho U^2$)

6.66 Turbulent Kinetic Energy

The centreline values of turbulence intensity ($\frac{[u'^2]^{1/2}}{U_0}$) and

turbulent kinetic energy ($\frac{k}{\rho U_0^2}$) for three cases are displayed in

Figures 6.6-13 and 6.6-14. It is seen that the values decrease rapidly after the recirculation zones.

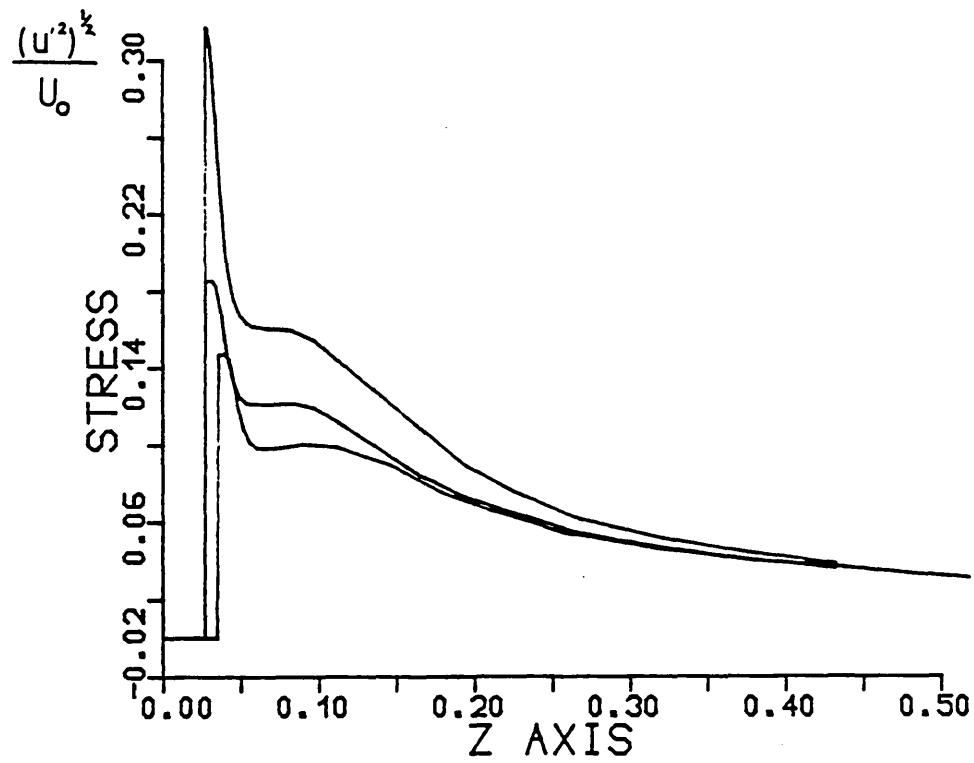


Figure 6.6-10 Centreline values of turbulence intensity for three lifts: (a) $L_V/D_V = 0.106$, (b) $L_V/D_V = 0.16$, (c) $L_V/D_V = 0.284$.

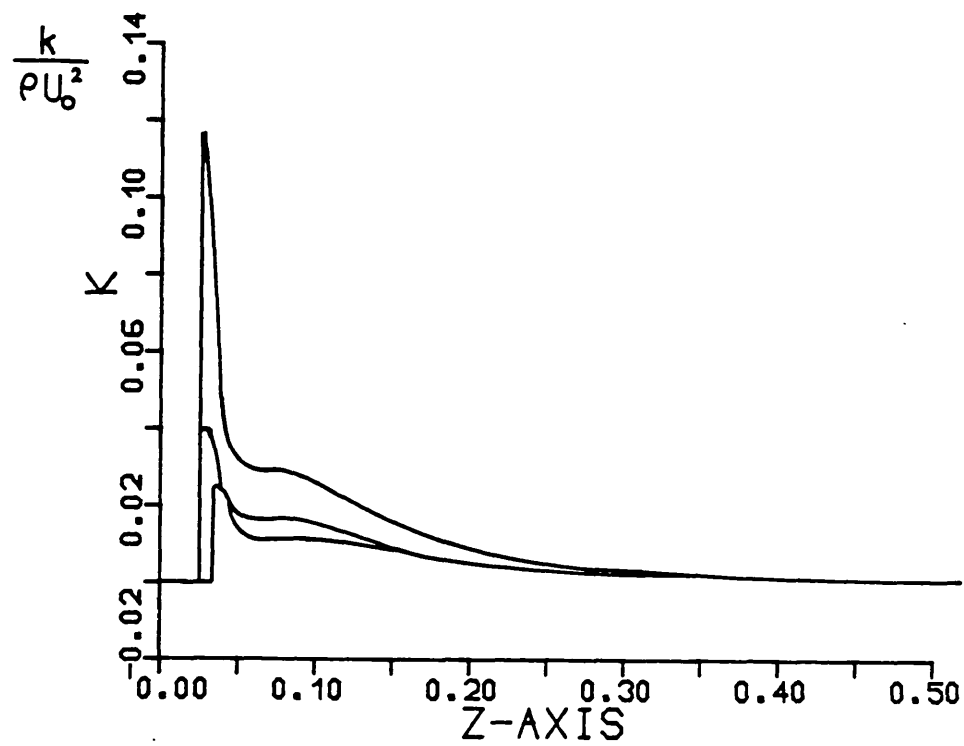


Figure 6.6-11 Centreline values of turbulent kinetic energy for three lifts: (a) $L_V/D_V = 0.106$, (b) $L_V/D_V = 0.16$, (c) $L_V/D_V = 0.284$.

6.67 Discharge Coefficient

(1) Effect of the Valve Lift on the Discharge Coefficient

(Cases 2-8)

Figure 6.6-15 shows the variation of the predicted discharge coefficient C_D for a valve with a 45 deg seat-angle and $W/D_v = 0.035$, with the dimensionless valve-lift (L_v/D_v). The results are also represented in Table 6.6-2. It is seen that, in general, higher C_D values appear at low lifts, with the maximum at $L_v/D_v = 0.1$, after the peak, the discharge coefficient decreases with lift.

(2) Effect of the Seat-Width on the Discharge Coefficient

(Cases 9-15)

Figure 6.6-16 presents the results of another series of calculations for cases 9 to 15, similar to cases 2 to 8, but with a wider seat width. It can be seen that at low and intermediate lifts, the discharge coefficients of a wider seat valve ($W/D_v = 0.068$) are in general smaller than those of a narrower one, and there is a dip for the wider seat ($W/D_v = 0.068$); a similar observation was also reported by Annand (A much deeper dip appeared in his experiment).

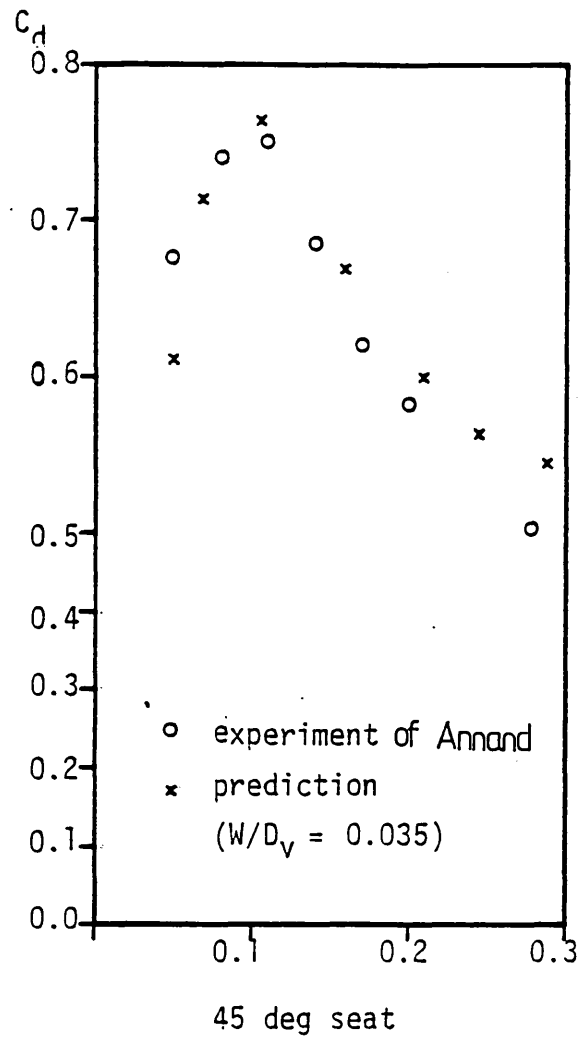


Figure 6.6-12 The effect of the lift on the discharge coefficient.

Table 6.6-2 Discharge coefficient for flow through a valve with 45° seat.

$W/D_V = 0.035$.

W/D_V	0.035						
θ	45°						
L_V/D_V	0.053	0.071	0.106	0.16	0.213	0.248	0.284
C_D	0.6033	0.7176	0.7752	0.6789	0.6062	0.5753	0.5547

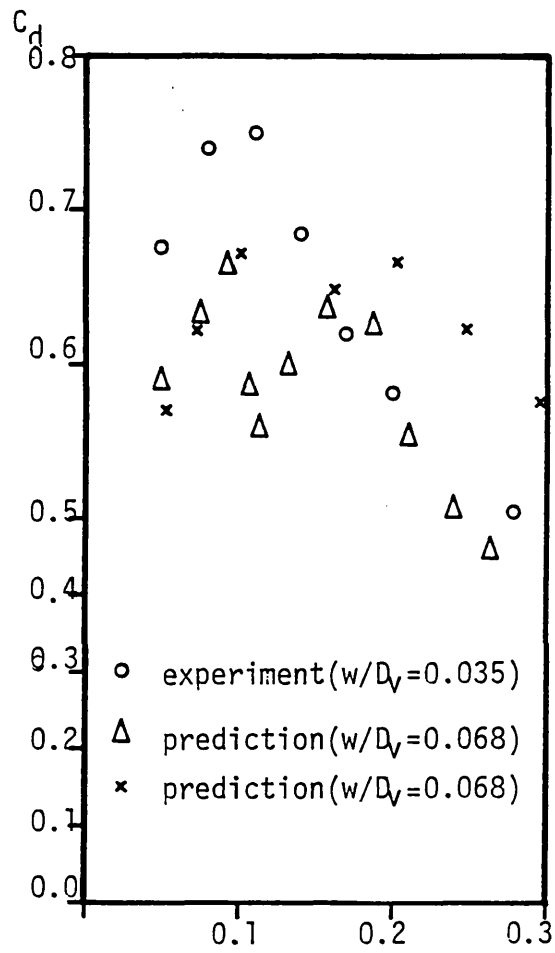


Figure 6.6-13 The effect of the seat width on the discharge coefficient.

Effect of the Seat Angle on the Discharge Coefficient

(Cases 16-21)

In Figure 6.6-14 is shown the calculated discharge coefficients for a number of different valve-seat angles, for the same seat-width and lift ($W/D_v = 0.035$, $L_v/D_v = 0.16$). The results are also represented in Table 6.6-3.

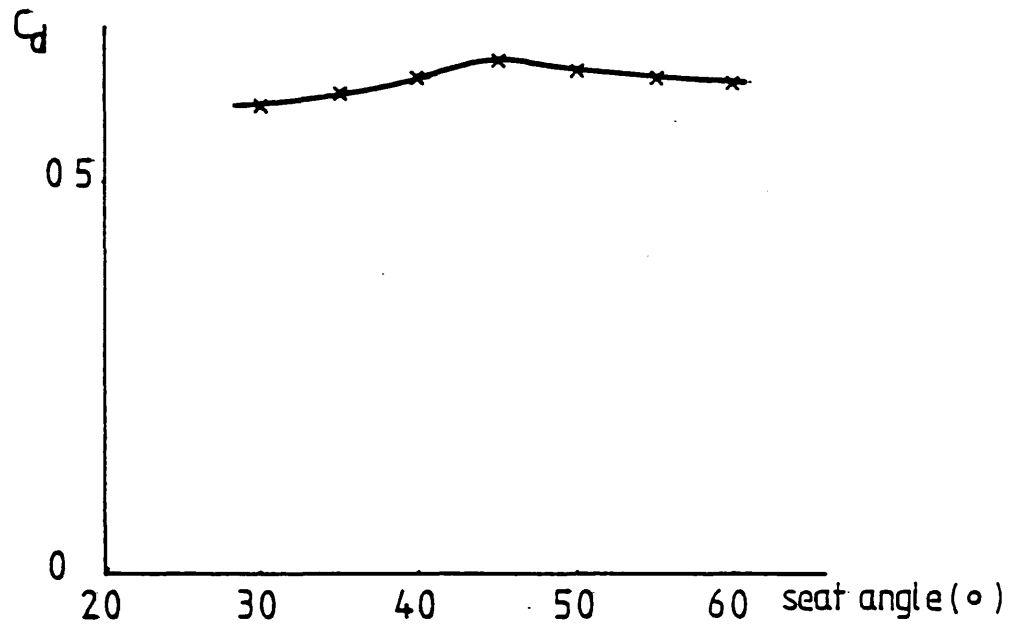


Figure 6.6-14 The effect of the seat angle on C_D .

Table 6.6-3

W/D_V	0.035					
θ	30°	35°	40°	45°	50°	60°
C_D	0.6090	0.6237	0.6429	0.6789	0.5586	0.6468
L_V/D_V	0.16					

6.7 Discussion

The aspects of the flowfield that are of interest in this study are the profiles of axial and radial velocity components, the wall pressure, the characteristics of the recirculation zone, and the level of turbulence which are generated. The effect of the valve lift on the above characteristics, and the influence of other geometry parameters of the valve such as valve seat angle, seat width on the global parameter, e.g. the discharge coefficient, are examined in the following subsections.

6.71 The Characteristics of the Flow Field Velocity and Pressure

The predicted velocity distributions at several streamwise locations, shown in Figures 6.6-2 to 6.6-4, have displayed a similar flow structure for three dimensionless valve lifts, namely 0.106, 0.16 and 0.284. The structure also resembles that for a seatless valve as presented in chapter 5.

The centreline profiles of axial velocity represented in Figure 6.6-5 show that the maximum negative velocity decreases with increasing valve lift. The maximum negative centreline axial velocity is interesting, because it can be regarded as a characteristic velocity of the recirculation zone, and also because it is related to the magnitude of the recirculating mass flow rate. Table 6.6-1 compares values of these quantities; and shows that the smaller valve lift is associated with a larger maximum mass flow rate of reverse flow, and larger length of recirculating zone.

The reason for the larger maximum centreline velocity associated with a small valve-lift may be explained as follows:

The flow has a larger radial velocity component at the valve exit for a smaller lift, see Figure 6.6-6, results in a wider recirculation zone, a sharper streamline curvature, a narrower gap round the recirculation zone, and a smaller pressure minimum at the 'eye' of the recirculation zone. Figure 6.6-9(a) shows that a sharper pressure gradient is associated with a smaller lift. The maximum reversed velocity is dependent on the centreline pressure gradient. As a result, the maximum negative centreline velocity is also larger.

6.72 Turbulence Quantity

The centreline values of turbulence intensity and turbulent kinetic energy depicted in Figures 6.6-10 and 6.6-11 for three valve lifts (L_v/D_v) of 0.106, 0.16 and 0.284, reveals qualitatively similar distribution. This result is unsurprising because similar qualitative velocity fields lead to similar rates of strain. It is also found that large $k/\rho U_0$ and $(u^2)^{1/2}/U_0$ values are located behind the valve because the recirculation there generates intense turbulence. The influence of the lift on the turbulence energy can also be seen from Figure 6.6-11: the smaller the lift, the larger the maximum turbulence energy. The reason is as follows: a smaller lift gives rise to a larger recirculation zone, with a corresponding sharper streamline curvature, particularly downstream of the maximum recirculation zone. It results in a smaller minimum pressure at the 'eye'; and corresponding smaller reversed velocities at the centreline

with a larger value of velocity gradient, $\frac{\partial U}{\partial z}$. This increases the

normal-stress-normal-strain interactions, and, therefore, gives a larger turbulence kinetic energy.

6.73 The discharge Coefficient

The variation of the discharge coefficient, C_D , with valve lift under the same Reynolds number is shown in Figure 6.6-12, and reveals that the C_D values are generally high for smaller valve-lifts and exhibits a peak at about $L_v/D_v = 0.1$, suggesting that the flow remains close to the walls of the valve passage and any separation occurring in the passage is small in the low valve-lift range. The decrease of C_D in the intermediate-to-high lift range is caused by the flow separation near the tip of the valve head. They are comparable with the experimental data of Annand. For an intermediate lift range, the prediction seems to be in fair agreement with the experiment. However, in general the predictions under-estimate C_D at small values of L_v/D_v and over-estimate it at high lifts.

The results for a wider valve, ($W/D_v = 0.068$) shown in Figure 6.6-13, display that at low lift region, the flow remains attached to the valve boundaries and fills the entire exit plane resulting in high values of discharge coefficient with a peak at $L_v/D_v = 0.09$. The values, C_D , falls with a dip. It might be caused by the separation occurring at the valve face. At intermediate valve lifts, a gradual improvement in the discharge coefficient is evident. At high lift, C_D decreases with increasing lift. It is found that the C_D values are higher for a thinner valve-seat than those for a wider one. The decrease of discharge coefficient is caused by the separation. It is, therefore, possible to improve the breathing capacity by a careful design of the valve configuration. Improvement can be achieved by a thinner valve and rounding off the sharp edges in the valve passage to minimise the possibility of flow separation.

The predictions are also compared with Annand's data. At low and intermediate lifts, the predictions appear to be in agreement with the experiment, but with a poor agreement at large lifts (the maximum discrepancy between prediction and experiment is above 10%).

The effect of the seat angle on the discharge coefficient can be seen from Figure 6.6-14. The predicted maximum value of C_D is obtained for a 45° seat angle valve

6.8 Closure

It has been demonstrated in this chapter that flow through a valve with an angled seat can be predicted satisfactorily, and the effect of three geometric parameters, i.e. valve lift, valve seat width and seat angle, on the flow has been studied. The discharge coefficient of flow is compared with the experiments of Annand.

The following influence of the valve lift on the flow has also been found: the smaller valve lift is associated with a larger maximum negative centreline axial velocity, a larger recirculating mass flow rate, and a lower minimum wall pressure.

CHAPTER 7 – THE EXPERIMENTAL INVESTIGATION: FLOW OF WATER INTO A CYLINDER THROUGH A COAXIAL VALVE-PORT

7.1 Introduction

This chapter is concerned with an experimental investigation of the laminar flow through an idealized valve geometry. For this study, a water-model rig was constructed in such a way that details of the flow pattern near the valve can be readily visualized and photographed. The choice of water as the fluid medium has been made for two basic considerations. Firstly, by using water it is possible to use a low velocity in the experiment to achieve a given Reynolds number; secondly, and perhaps more importantly, dyed fluid is more effectively used with water.

The purpose of this chapter can therefore be summarised as follows:

- (a) To observe the details and note any unusual behaviour of the flow pattern around the valve;
- (b) To obtain photographic records of the flow patterns for steady, as well as unsteady flows through a coaxial valve-port for comparison with computer predictions.

The remainder of this chapter is divided into seven sections. Section 7.2 contains a description of the apparatus used and the operational procedure. In section 7.3, the various cases studied are detailed. Observations and discussion of steady and unsteady flow tests are contained in section 7.4 and 7.5 respectively. Section 7.6 describes an

unexpected circumferential motion phenomenon. The last section, section 7.7 summarises what has been achieved in this experimental investigation.

7.2 Description of the Experiments

7.21 Apparatus

The general arrangement of the apparatus used for the present series of experiments is schematically depicted in Figure 7.21-1. (the main dimensions are given in Figure 7.21-2). Water from a constant-level supply tank, 2, flows past a poppet valve, 10, into the test cylinder, 12, via the pipes 7 and 9. When the water in the cylinder is full, it overflows into a rectangular perspex container, 14, where the water level is kept constant by the overflow, 13. This rectangular-box arrangement allows the flow inside the cylinder to be visualized with a minimum of distortion. The water-supply tank and the test-cylinder assembly are mounted on a metal framework about 2.5m and 1.5m respectively above the floor, giving a maximum head of 1.35m for the flow. Inside the test cylinder, the poppet valve, 10, attached to a support, 17, is held coaxially with the cylinder at a short distance above the water inlet, this distance represents the valve lift and can be adjusted through a nut-and-thread arrangement.

Figures 7.21-2a and 2b show the geometrical details of two idealized valves used, the seatless one is shown in Figure 7.21-2(a) and the other, with a seat angle of 45° , in Figure 7.21-2(b). The dimensions of the valves tested are given in table 7.21-1.

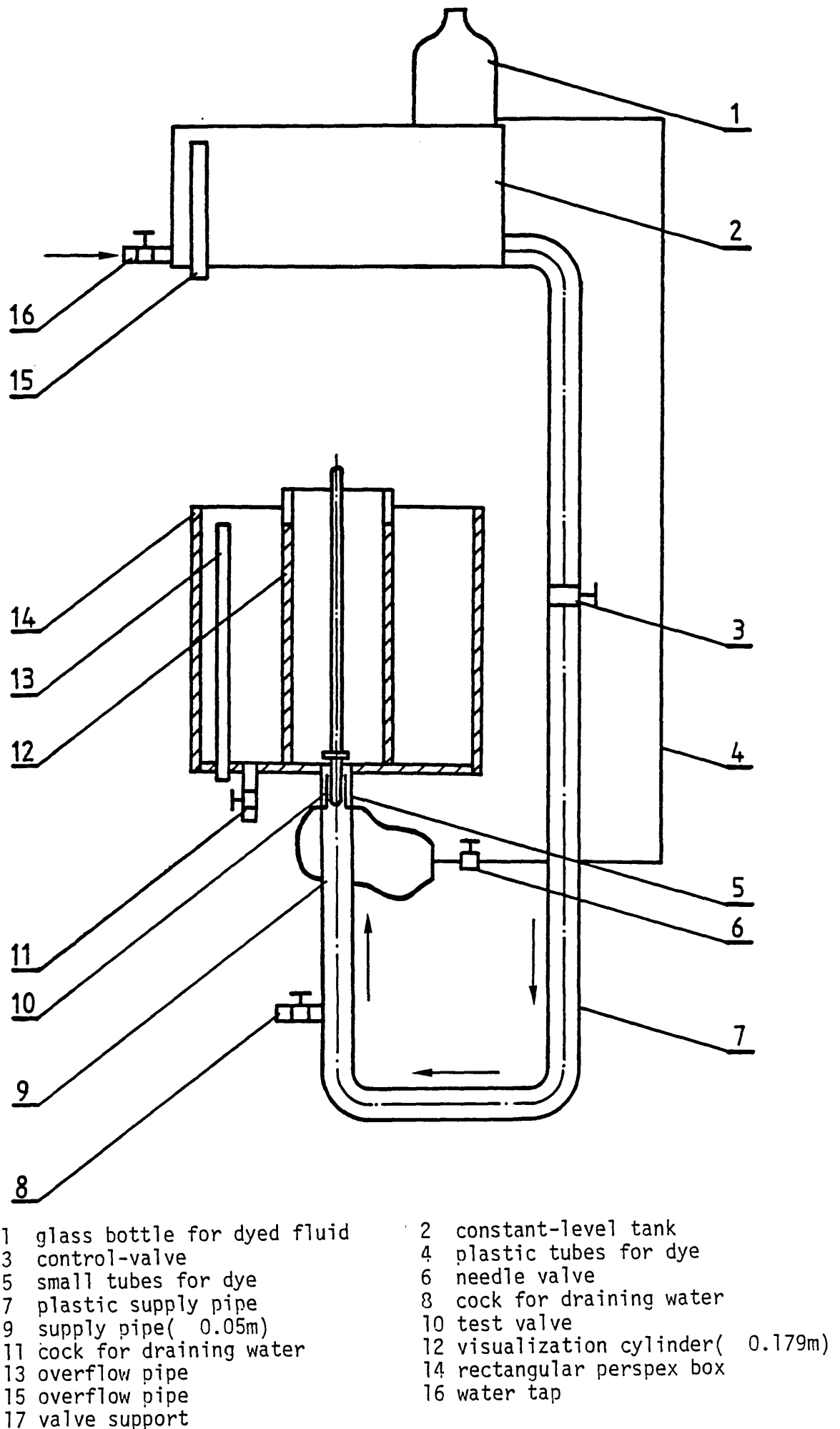


Figure 7.21-1 Experimental apparatus .

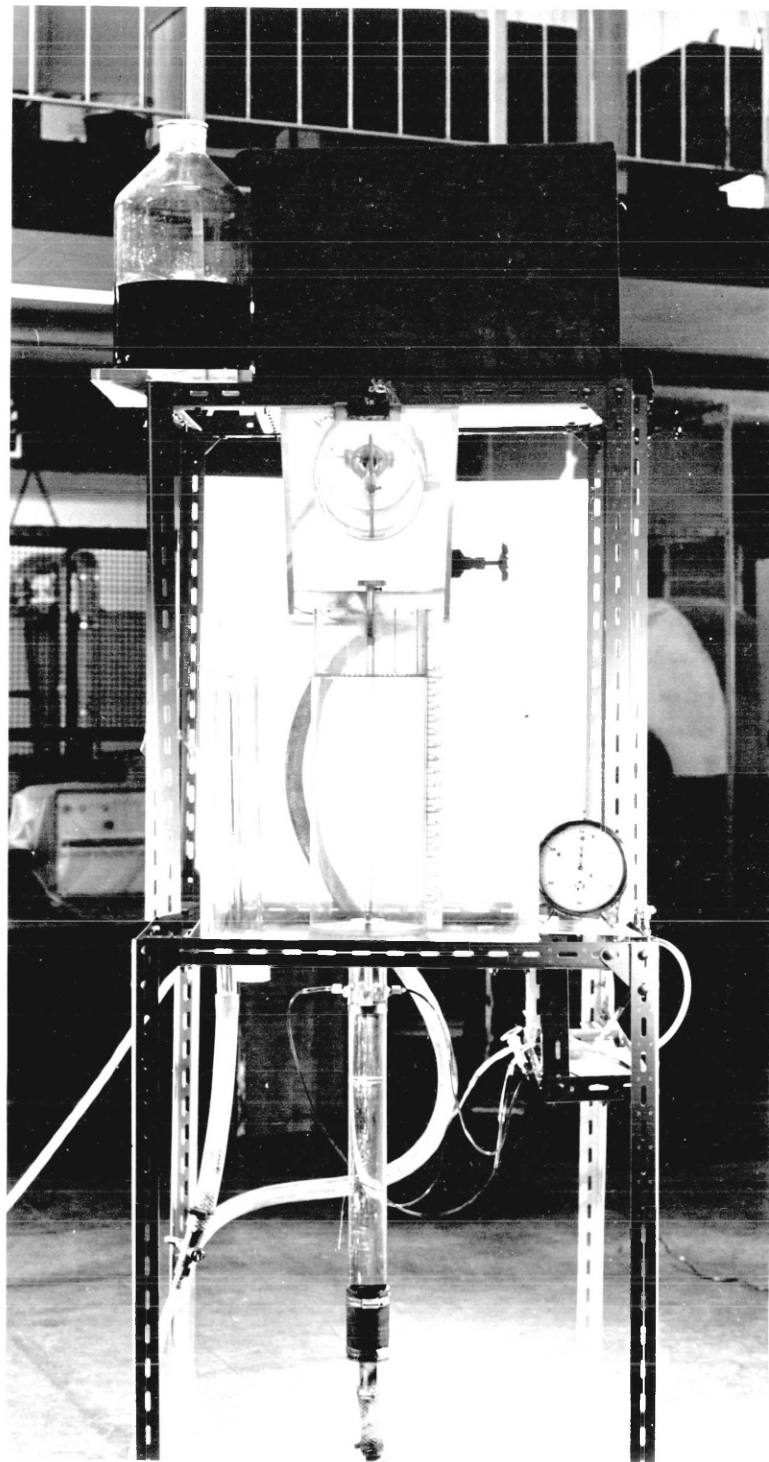
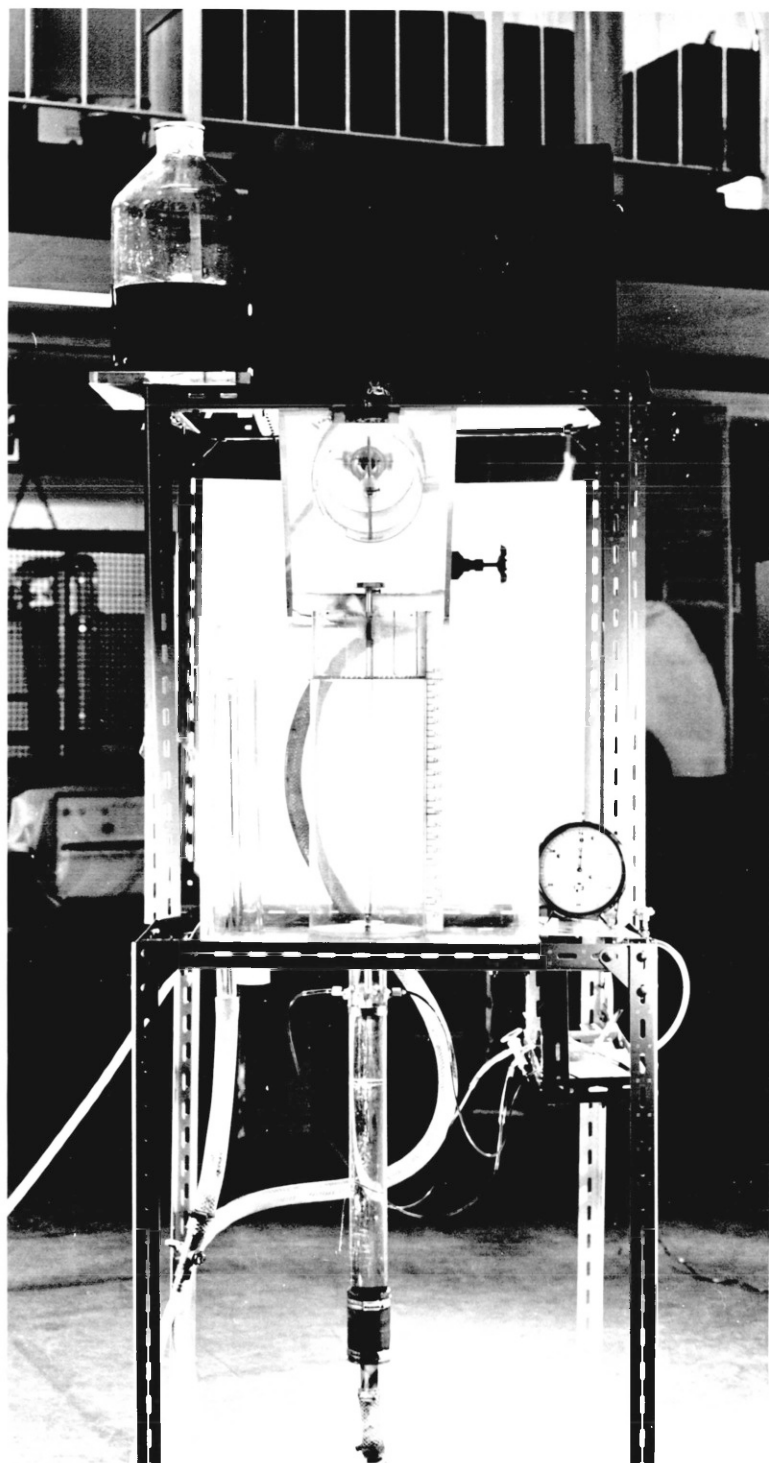
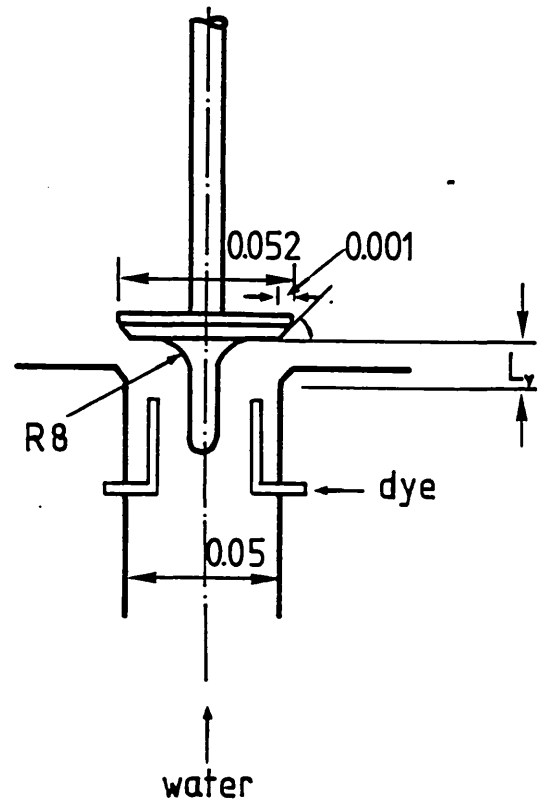


Figure 7.21-1(b) Photograph of the experimental rig.





(b) seated valve

Figure 7.21-2 Intake valve-port geometry used in the experiment.

Table 7.21-1 Dimensions of valves tested

	seatless valve	valve with a seat angle of 45°
valve head thickness	0.002m	0.002m
valve head diameter	0.05m	0.052m
seat angle	0	45°
seat width	0	0.001m

The water flow-rate is measured by timing the change in water level in the cylinder.

The flow around the valve is made visible by injecting a small amount of dyed fluid from the glass bottle above the tank through two hypodermic tubes, 5, as shown in Figure 7.21-1. These tubes are each of 0.002m diameter and are placed inside the supply pipe with their exits at a short distance (0.009m) below the cylinder bottom. The rate of injection of the dyed fluid can be adjusted by a needle valve in the flow circuit, 4. The dye used is Nigrosine; the density of the dyed fluid is approximately equal to that of the water (within 0.1%).

Two halogen lamps are used as the light source for the experiment. A sheet of translucent polythene paper is attached to the back of the rectangular box between the lamps and the test cylinder to create an even, diffused light background for photographic recording. The flow patterns are recorded both by a still camera and a video camera. For unsteady flows, time-reference marks are provided by an electric clock.

A mirror which is placed above inclined at 45° to the water level is used to give a top view of the experiment, as illustrated in Figure 7.21-3. This arrangement can be used for both steady and unsteady tests.

7.22 Procedure

A selected poppet valve is fitted on to the support, and the valve lift is adjusted to the desired distance, which is then kept constant during any one entire experiment. The rectangular perspex box surrounding the test cylinder is then filled with water to a height of 0.35m. The mirror is adjusted to a suitable position. Further details vary somewhat according to whether the flow is a steady one or not. The description of these is divided into two parts as follows.

(a) Steady flow

Once the control valve is opened, water from the tank flows through the pipes and past the poppet valve into the test cylinder. When the water level reaches holes, which are situated along the cylinder wall at a height of 0.35m, it overflows into the rectangular box.

The flow is then left to run for 5 minutes to attain steady-state conditions. At this point the needle valve is opened, continuously injecting the dyed fluid into the water through the two hypodermic tubes. The velocity of dyed fluid is carefully controlled so that it is nearly the same as that of the water to avoid any unnecessary disturbances to the flow.

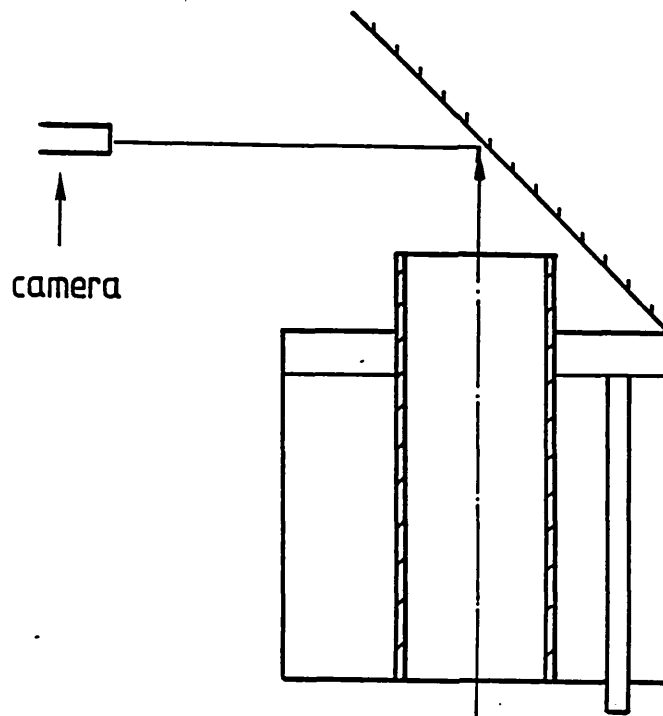


Figure 7.21-3 Arrangement of mirror used to the top view
(a) of the flow pattern.

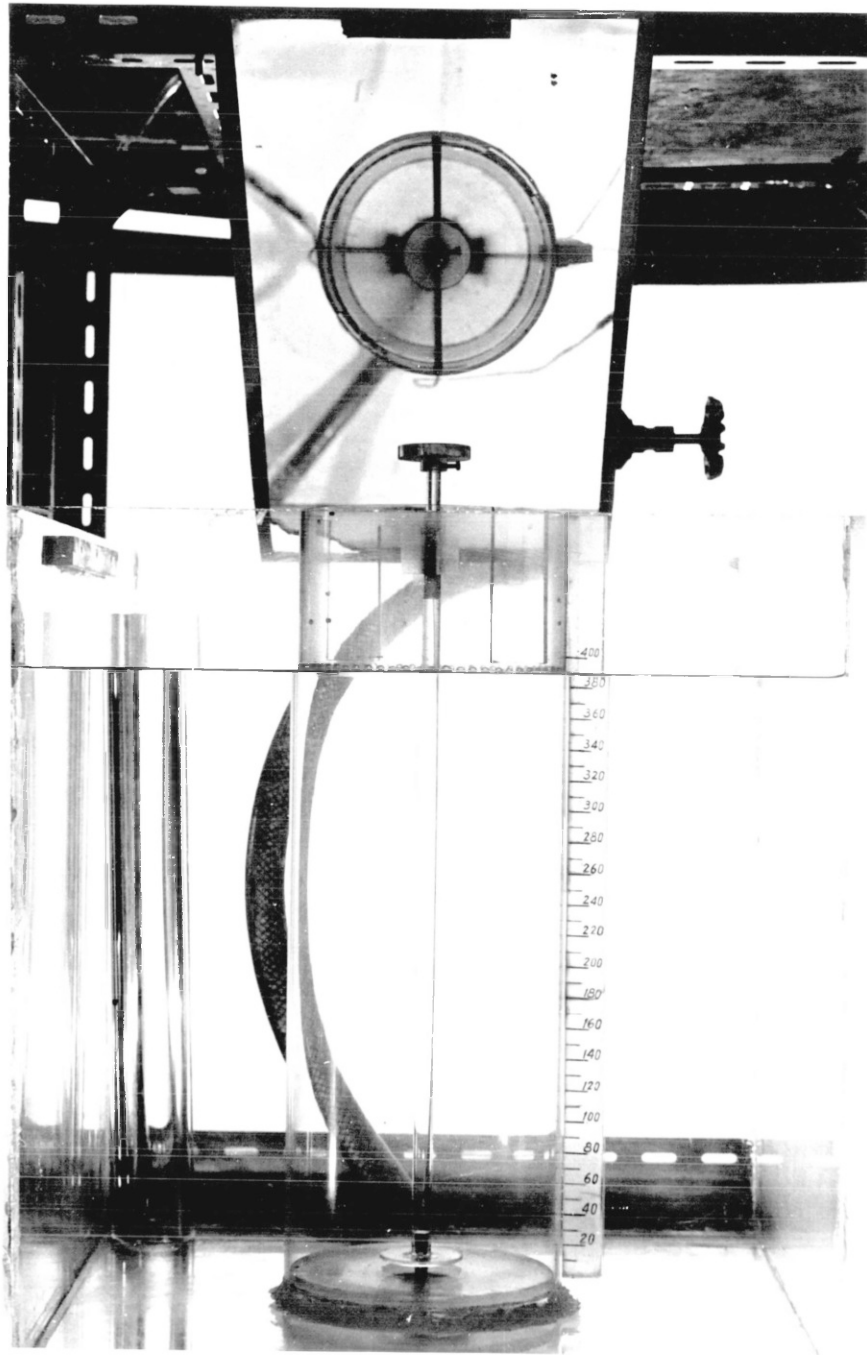
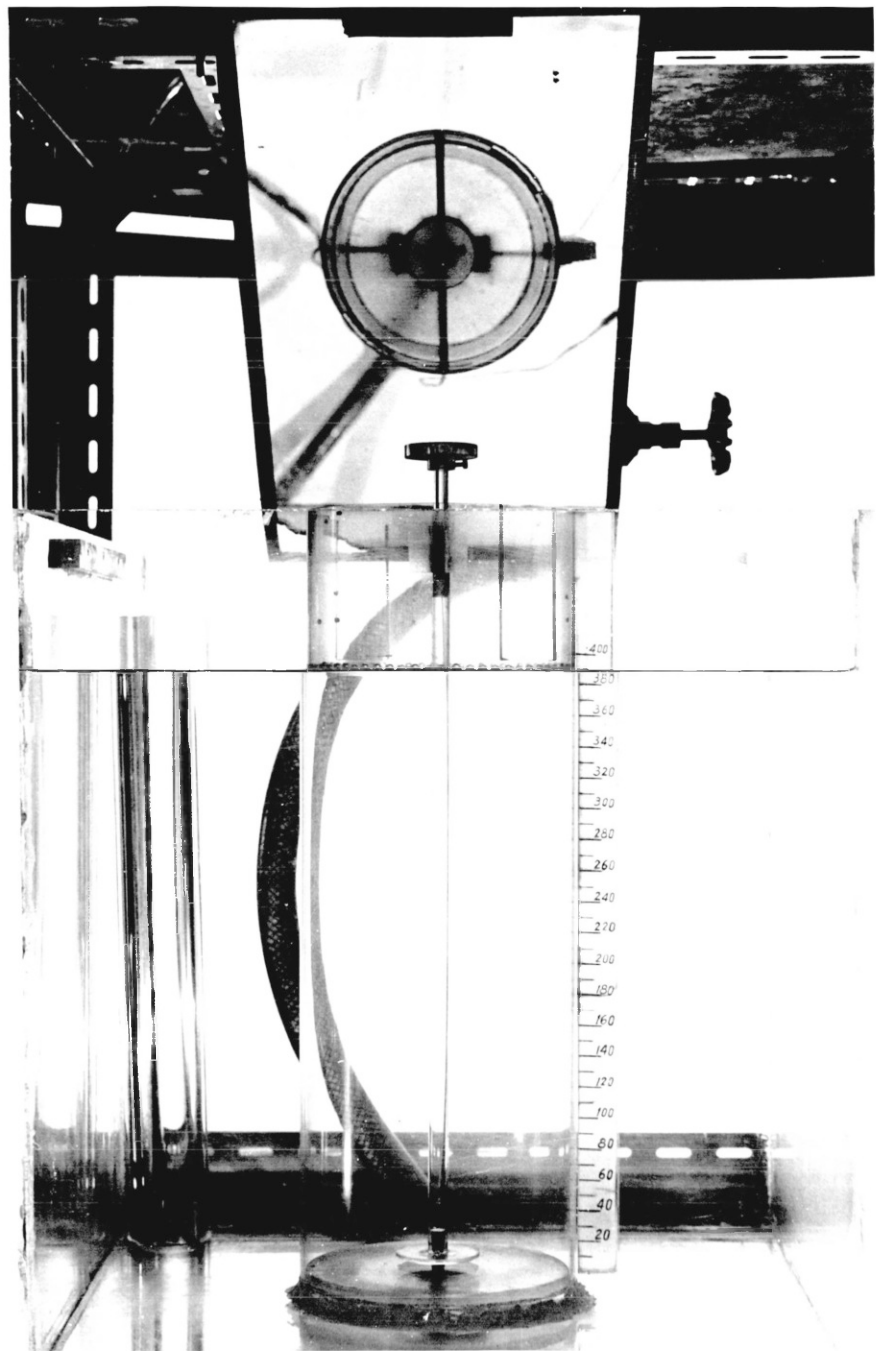


Figure 7.21-3(b) Photograph of the main part of the experimental apparatus.



The flow pattern around the poppet valve is then recorded by a still camera or a video tape recorder.

The mass-flow rate through the cylinder inlet is determined by timing successive water levels in the test cylinder as it fills up.

(b) Unsteady Flow

The poppet valve to be tested is assembled as described above. The test cylinder is initially empty. The control valve is opened to fill the cylinder with water at a pre-set rate. When the water level reaches 0.01m beyond (above) the poppet valve, two streams of dyed fluid are injected into the water. A series of flow patterns near the poppet valve are then recorded using either a still camera or a video recorder at specific time intervals.

The mass-flow rate is measured in the same way as for the steady flow.

To obtain a series of top views of the flow pattern, the experiment is then repeated with the camera focussed on the centre point of the cylinder as viewed from the mirror (see Figure 7.21-3)

7.3 Tests Carried Out

The test conditions in this series of experiments are chosen so as to allow the flow patterns around the valve to be studied under the influences of a range of parameters. In order to avoid unnecessary over-complications, a simple geometry with a stationary, axisymmetric intake valve is used. The flows in all tests are laminar, the range of

Reynolds number, based on the supply-pipe diameter and velocity, being between 400 to 1000.

Table 7.3-1 shows a summary of the experimental test conditions for both the steady and unsteady flows carried out. As can be seen, the influences of three of the parameters, Reynolds number, Re , valve lift, L , and the valve seat angle, θ , have been investigated. Valve lift is here defined as the axial distance between the position of the upstream valve-surface and its fully-closed position. The range of the dimensionless ratio valve lift/valve diameter, L/D_v , covered here has been chosen to fall within the range of this parameter commonly encountered in intake valves of IC engines.

Table 7.3-1 Summary of the experimental conditions

Test state	Test No	Reynolds number Re	Dimensionless valve lift L/D_v	valve seat angle θ	stem diameter (m)	valve diameter (m)
steady flow	1	400	0.14	0°	0	0.002
	2	620	0.14	0°	0	0.002
	3	1000	0.14	0°	0	0.002
	4	620	0.14	45°	0.008	0.002
unsteady flow	5	620	0.14	45°	0.008	0.002
	6	400	0.14	45°	0.008	0.002
	7	1000	0.14	45°	0.008	0.002
	8	620	0.08	45°	0.008	0.002
	9	620	0.2	45°	0.008	0.002
	10	620	0.24	45°	0.008	0.002
	11	620	0.14	0°	0.0	0.002

7.4 Steady Flow Tests - Observations and Discussion

Figures 7.4-1 to 7.4-3 show the flow patterns observed near a seatless valve under steady and axisymmetric conditions for three Reynolds numbers, 400, 620 and 1000 respectively, (i.e. tests 1, 2 and 3 in Table 7.3-1).

They indicate clearly the presence of a structured flow with two distinct vortices: the smaller one is situated in the corner of the cylinder and the other large one just downstream of the valve. The upper vortex on the left-hand side and the lower on the right-hand side rotate in a clockwise direction; the upper vortex on the right-hand side and the lower one on

the left-hand side rotate in an anti-clockwise direction. The upper vortex is formed due to the obstruction of the coaxial disk-shaped valve by rolling up inwards of the boundary layer on the cylinder wall. The lower is formed due to the pipe expansion and rolls down outwards. These vortices trace the pathlines of the dye fluid particles, originating from the two point sources just below the inlet (see Figure 7.21-2). Although they are toroidal, the flow patterns seen on the photographs can thus be taken as the streamlines in a diametrical plane through the axis of the cylinder. Once the vortices have been formed, their positions and sizes do not change significantly throughout the experiment.

Concerning the Reynolds number effect on the flow patterns, it is also seen that the traces of the vortices at a lower Reynolds number of 400 are the clearest and well defined. At a higher Reynolds number of 1000, the traces tend to break down into fragments and the dye diffuses more quickly. As a consequence the vortices appear to be less orderly and not so sharply defined; and they are not as stable as the one in lower Reynolds number test. At formation, the vortices are very clear but later on they become less defined and rise; and are replaced by new vortices, as shown in Figure 7.4-3. From Figures 7.4-1 to 7.4-3, it may also be observed that the larger is the Reynolds number, the smaller is the toroidal diameter of the vortex ring, and the larger, only slightly, the vortex tail angle (see Figures 7.4-1 to 3).

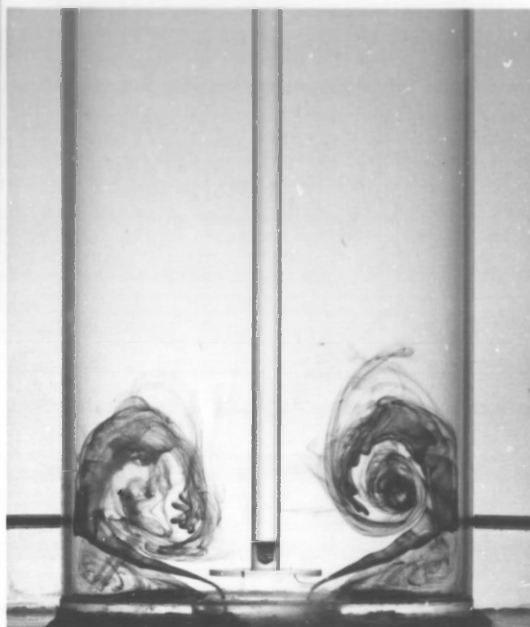


Figure 7.4-1 Steady flow test:
 $Re=400, L/D=0.14, \theta=0^\circ$.

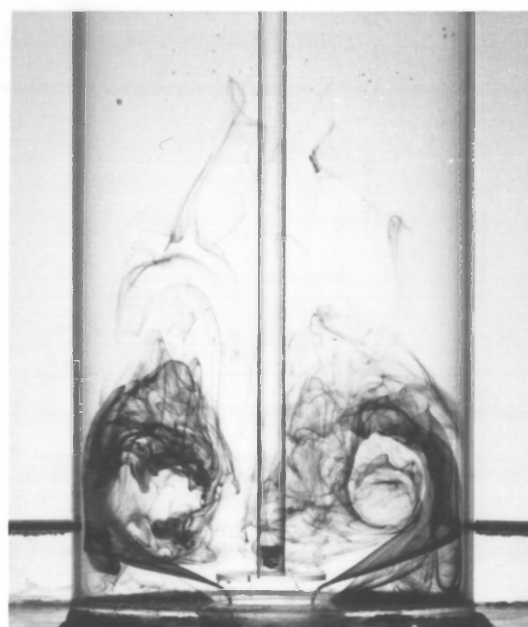


Figure 7.4-2 Steady flow test:
 $Re=620, L/D=0.14, \theta=0^\circ$.

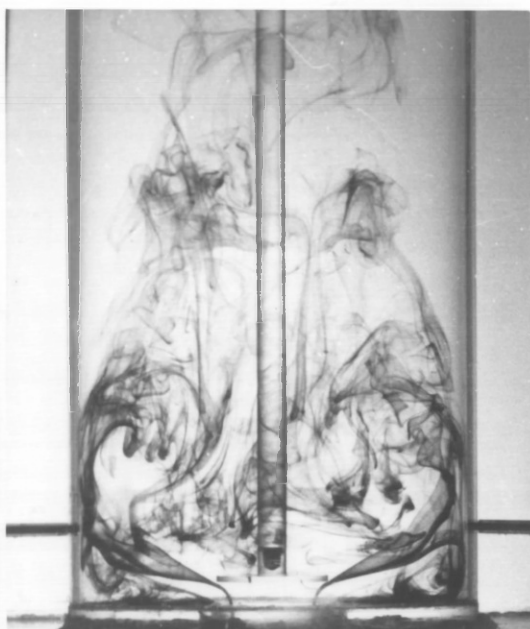


Figure 7.4-3 Steady flow test:
 $Re=1000, L/D=0.14, \theta=0^\circ$.

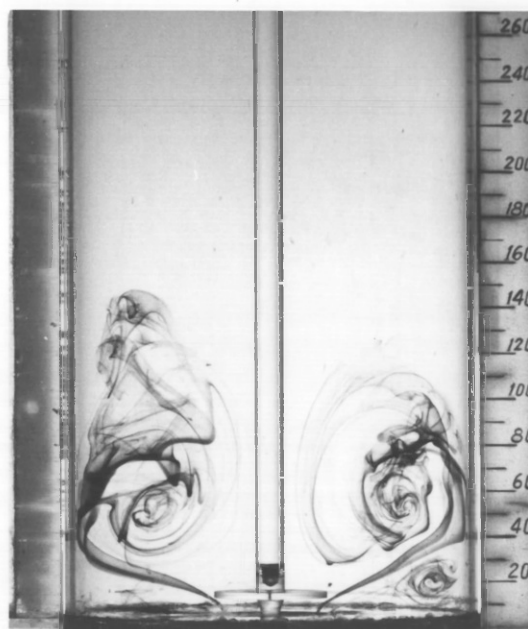


Figure 7.4-4 Steady flow test:
 $Re=620, L/D=0.14, \theta=45^\circ$.

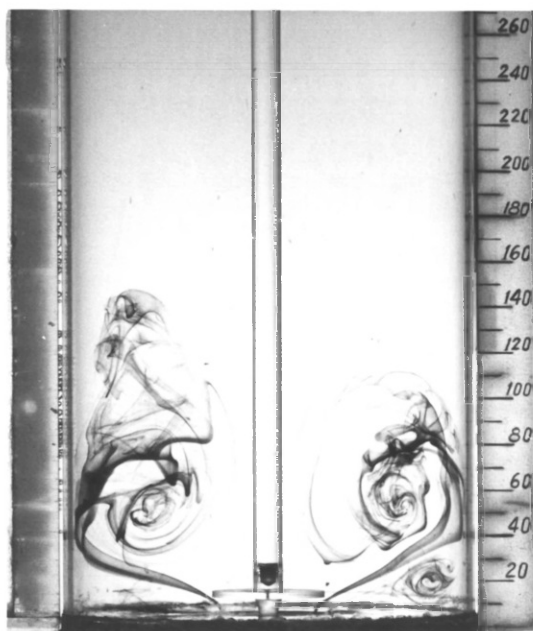
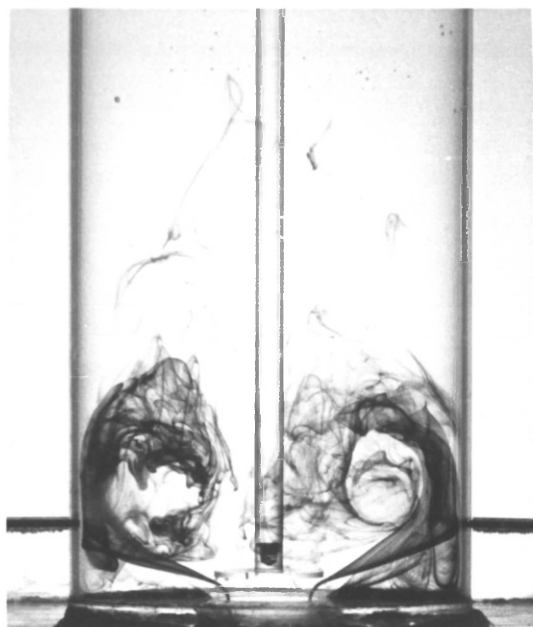
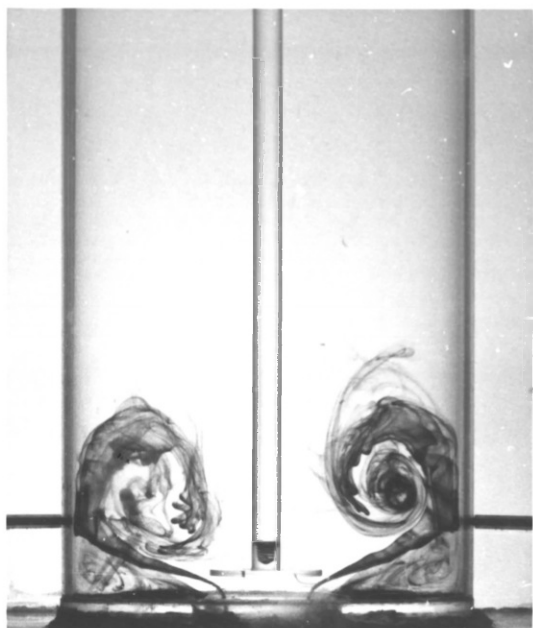
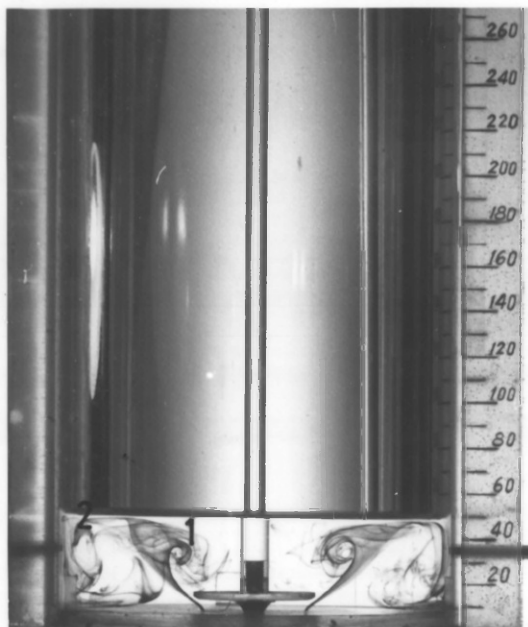


Figure 7.4-4 shows the flow pattern near a valve with a seat angle of 45° for a Reynolds number of 620 and a dimensionless valve-lift of 0.14 (i.e. Test No. 4 in Table 7.3-1). Compared with the corresponding flow in Figure 7.4-2, for a seatless valve, the position of the vortex core is higher than that of a seatless valve test case, and the tail angle of the vortex is larger than that of seatless valve case. (See Fig. 7.4-2 and 7.4-4) and the vortex shape is elongated because of the influence of the valve seat.

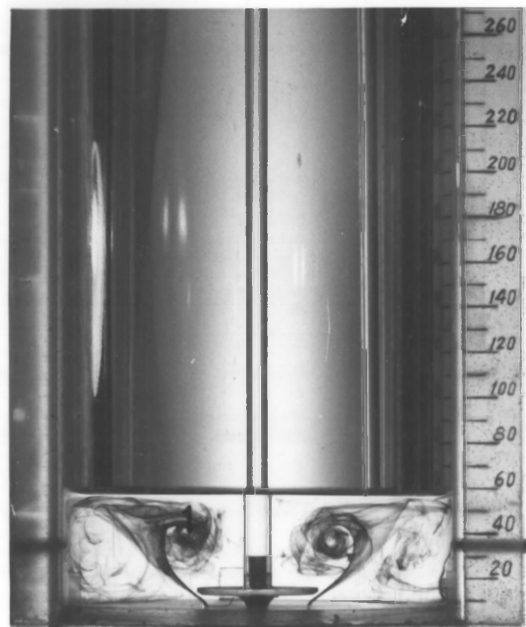
7.5 Unsteady Flow Tests - Observations and Discussion

Figures 7.5-1 to 7.5-7 show photographic records of several series of flow patterns around the valve while the cylinder was being filled slowly at a constant velocity. The effects of Reynolds number can be seen from Figures 7.5-1 to 7.5-3 (test No. 5, 6 and 7); and the effects of valve lift are displayed in Figures 7.5-4 to 7.5-7 (tests 5, and 8-10). The series of flow patterns in Figures 7.5-8 (test 11) are for a seatless valve; these may be compared with those of Figures 7.5-1 for one with a 45° seat angle.

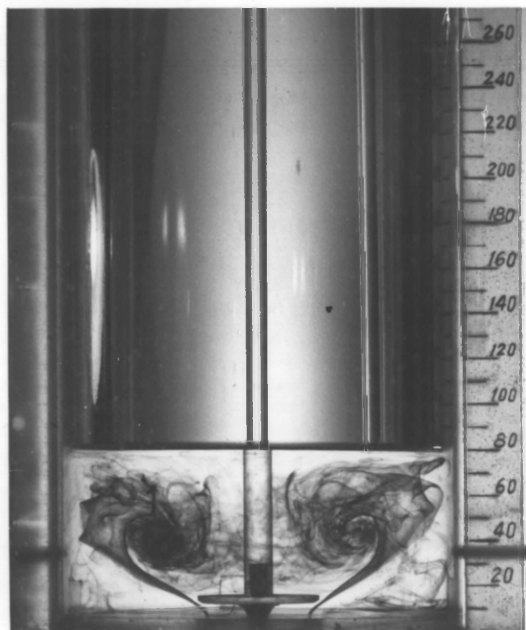
An apparent difference of stem diameter between the parts of the stem above and below the water level can be noted in all the pictures; this is due to optical distortion. Since the test cylinder is immersed in water in a rectangular container, the distortion arises because the stem above the water level in the test cylinder is surrounded by air.



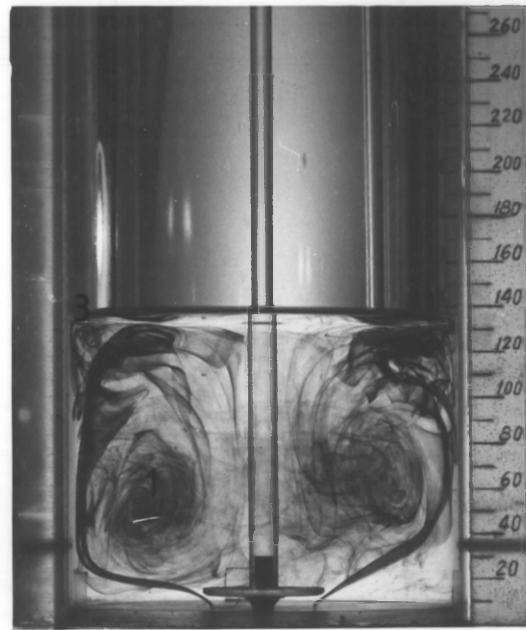
a. $t = 40$ seconds



b. $t = 48$ seconds

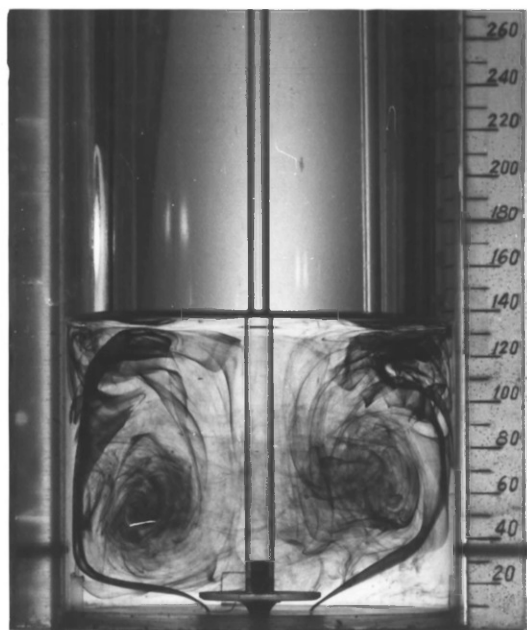
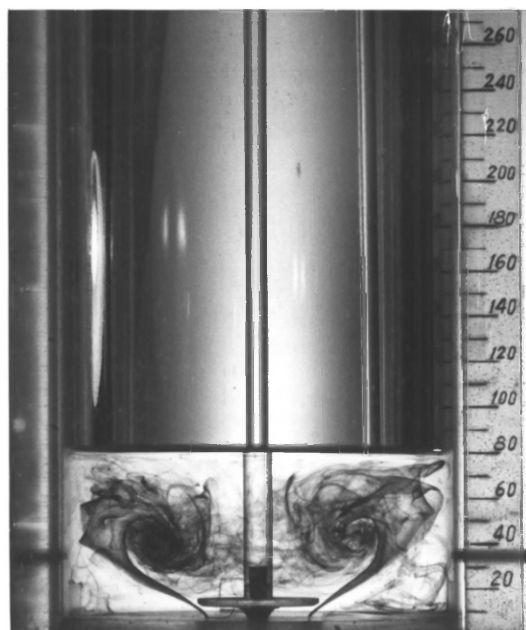
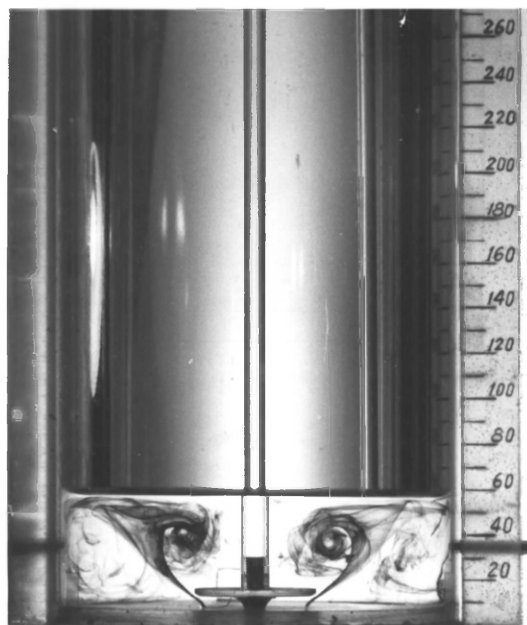
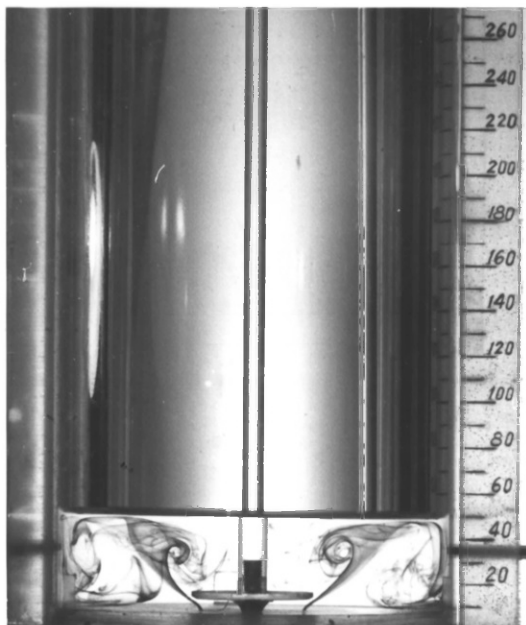


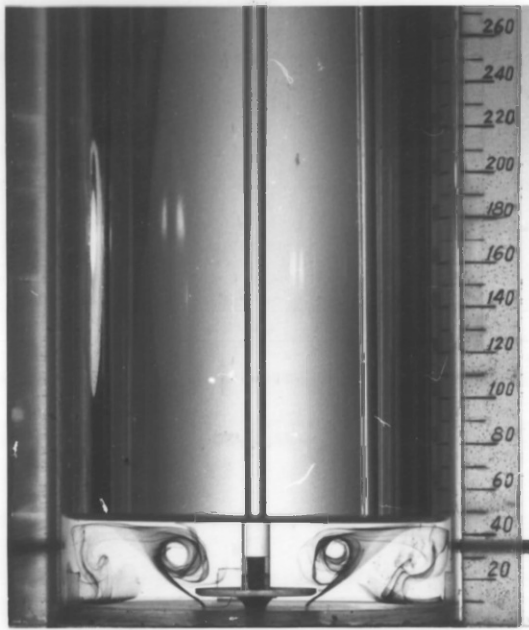
c. $t = 64$ seconds



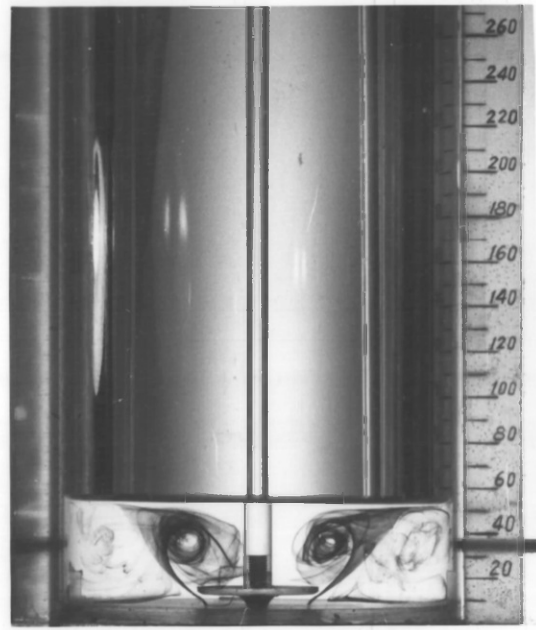
d. $t = 112$ seconds

Figure 7.5-1 Development of the flow pattern around a valve with seat angle of 45° ; $Re=620$, $L/D=0.14$.

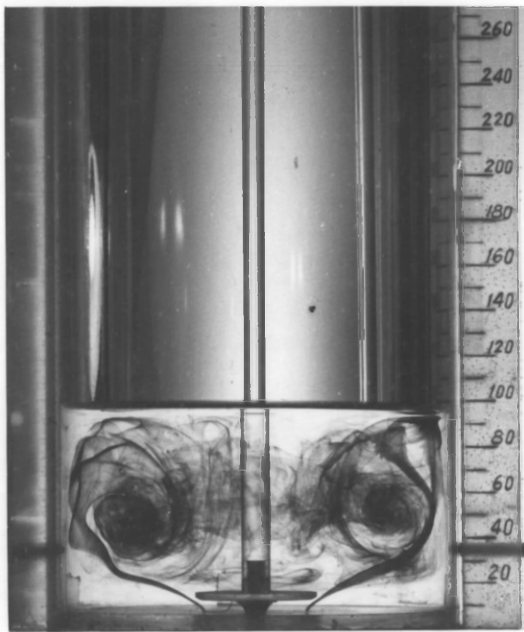




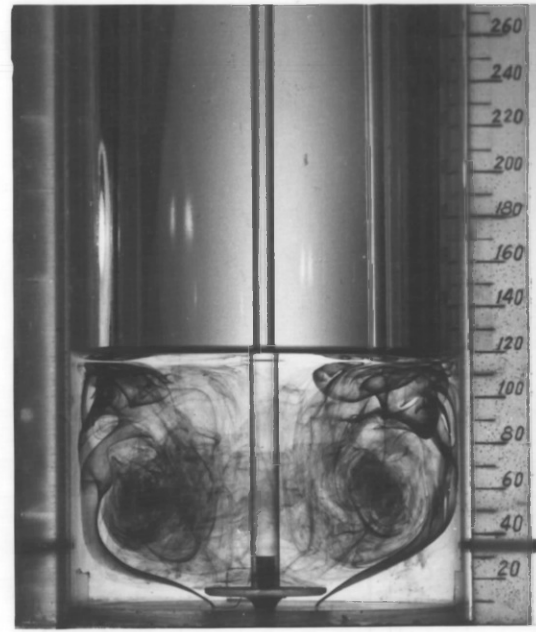
a. $t = 58$ seconds



b. $t = 63$ seconds

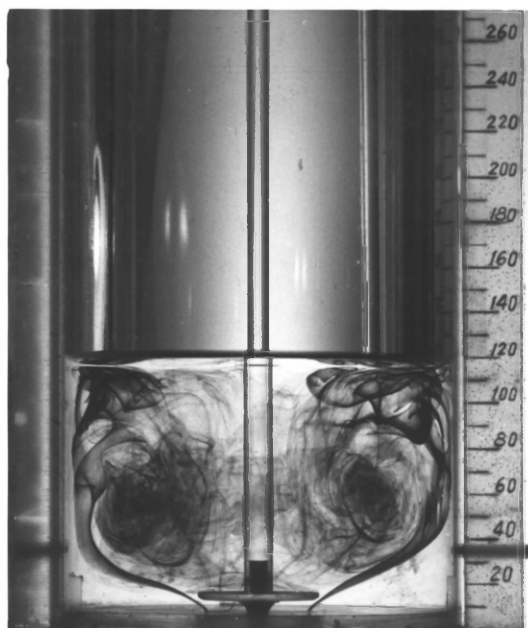
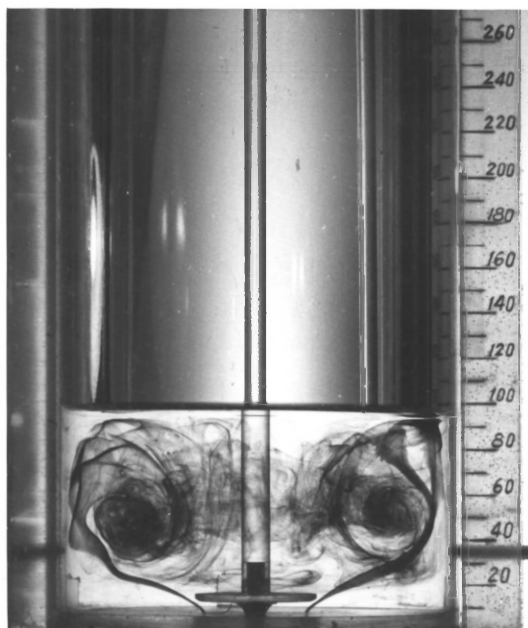
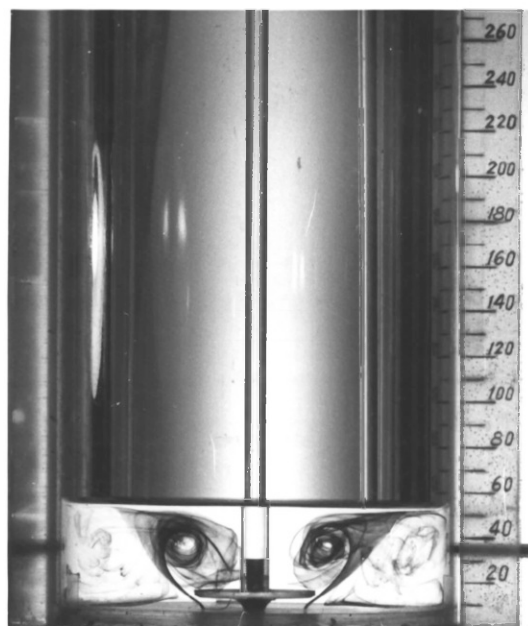
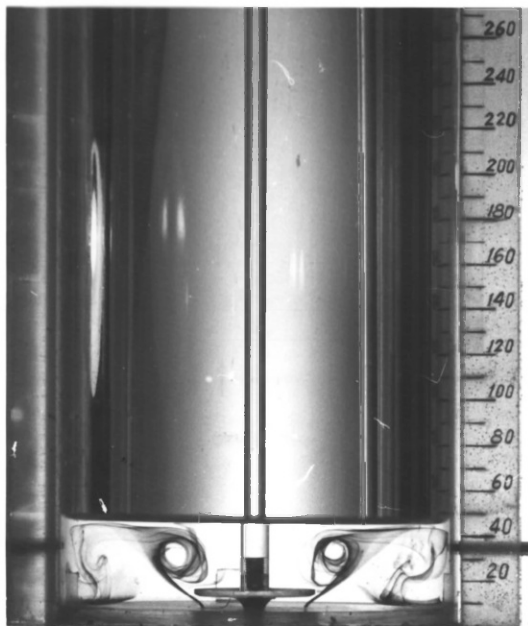


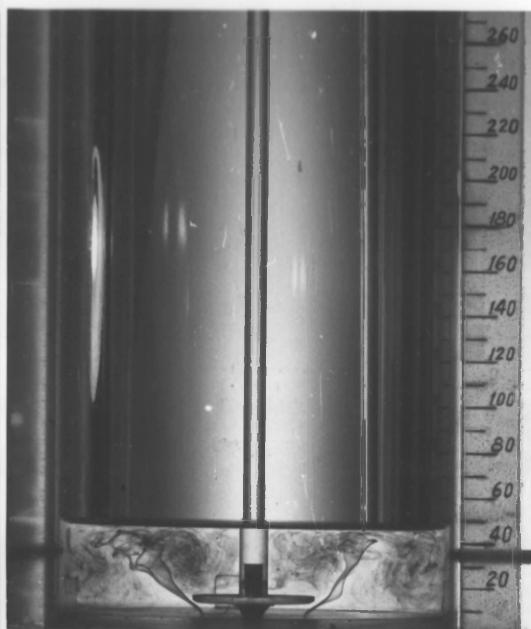
c. $t = 120$ seconds



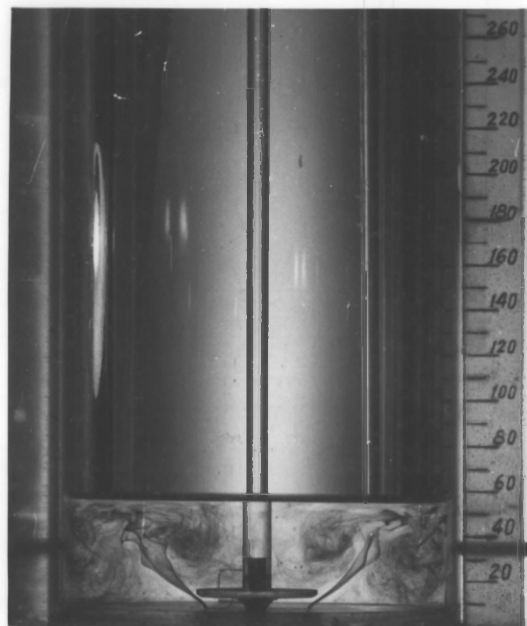
d. $t = 144$ seconds

Figure 7.5-2 Development of the flow pattern around a valve with seat angle of 45° ; $Re=400$, $L/D=0.14$.

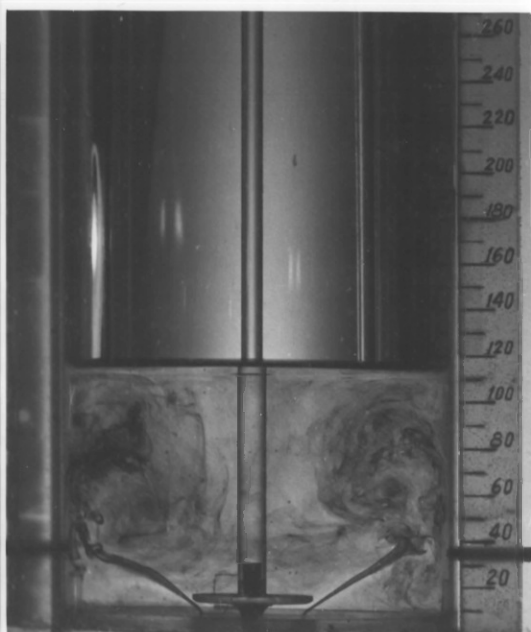




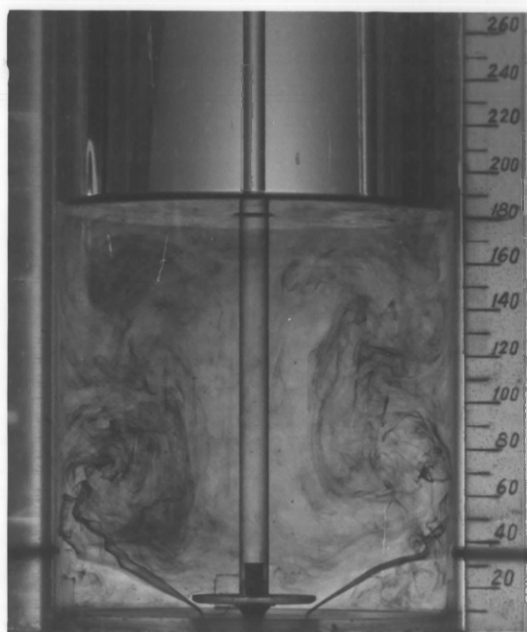
a. $t = 26$ seconds



b. $t = 32$ seconds

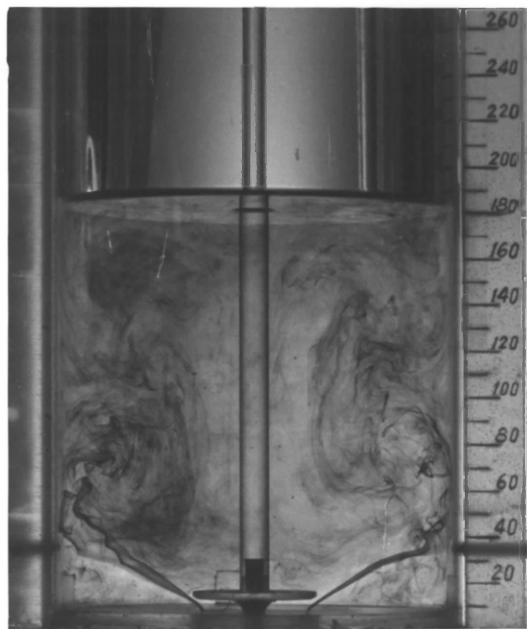
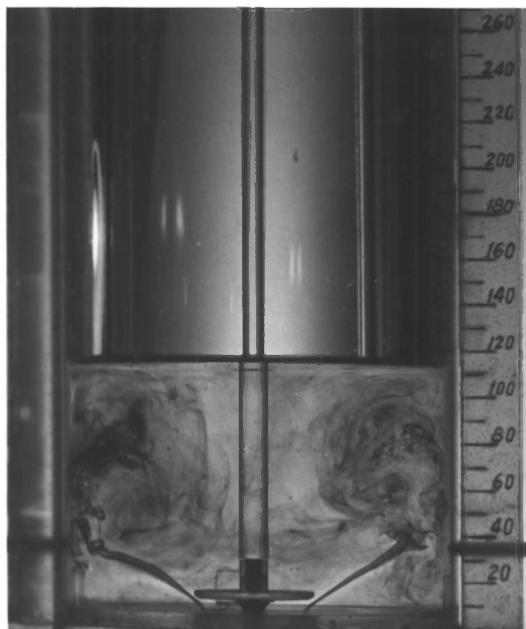
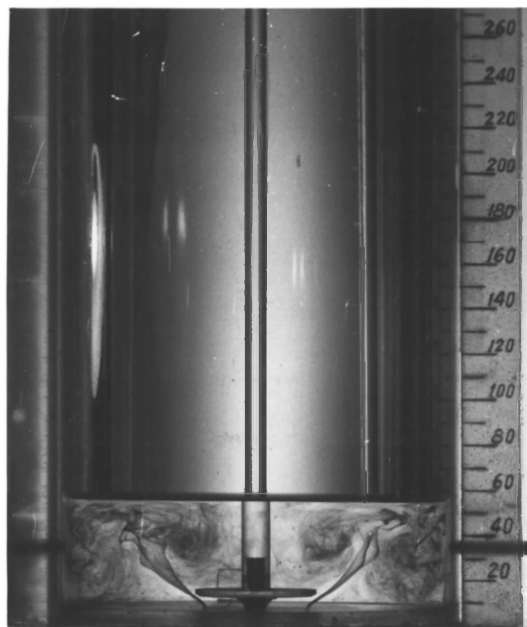
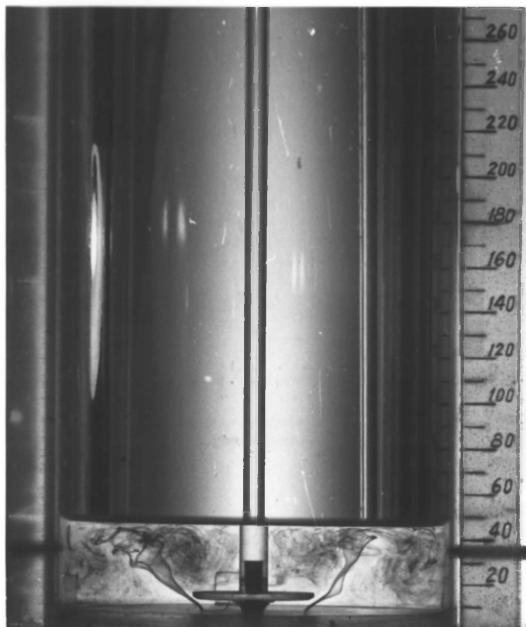


c. $t = 63$ seconds



d. $t = 102$ seconds

Figure 7.5-3 Development of the flow pattern around a valve with seat angle of 45° ; $Re=1000$, $L/D=0.14$.



The development of the flow-pattern around a valve with a seat angle of 45° , during the rising of the water at a steady velocity, can be seen from Figures 7.5-1 (a) to (d). The Reynolds number was 620 and the dimensionless valve lift was 0.14. Two distinct vortices can be clearly identified on the first photograph. One, marked 1, resulting from the obstruction of the disc-shaped valve, is situated just downstream of the valve close to the stem; it is restricted by the water surface as well as by the valve; the other, marked 2, near the bottom corner of the cylinder was formed as a result of the sudden expansion of the flow area at the inlet to the cylinder, and the effect of the water surface. The size of both these vortices was comparatively small at this early stage when the water level was low in the cylinder, 50mm, $t = 40$ sec. The second photograph, (b), shows the flow pattern at a water-level of 60mm, $t = 48$ sec., and indicates clearly the growth of the vortices. The main vortex, 1, had become larger, and the core of the vortex had moved radially outwards. The other one, marked 2, in the corner had become more diffused. Photograph (c) records the flow pattern at a water surface height of 80mm, $t = 64$ sec, the lower vortex in the corner, 2, was not affected by the water surface, and had become smaller, while the main vortex, 1, extended continuously with the rising water level; its core had moved further outwards. The fourth photograph, in Figure 7.5-1 (d), represents the developed flow structure at a water level of 140mm, $t = 112$ sec. The corner vortex had almost vanished because fresh dyed fluid no longer entered this region and the previously injected dye had diffused away. The main vortex, 1, fully developed, was situated at the certain position and no longer to change the position of its core. However, it had become darker because more dyed fluid had been injected. A third vortex, 3, now appeared just beneath the water surface near the cylinder wall; it rose with the rising water.

It should be noted that the real flow was three-dimensional in character, while all the photographs only show the flow patterns in or near the plane through the axis of the cylinder and the two dye-injection points.

Figure 7.5-2 shows a series of four flow patterns at a lower Reynolds number of 400. It can be seen that the dye continuously rolled up to form a vortex. Various stages of development of the vortices described for the flow at a Reynolds No. of 620 can also be identified in this series of photographs. In this test, the vortices were more sharply defined than those for the larger Reynolds number. The main vortex, 1, is seen to be smaller, because of the lower Reynolds number, and darker because the dye stayed longer in this region due to the lower flow velocity.

The series of flow patterns of Figure 7.5-3 is for a Reynolds number of 1000. It is seen that the dyed-fluid traces broke down into many fragments and the dye diffused more quickly; as a consequence the vortices appear to be less orderly and their shapes less well defined. It can also be seen that the position of the main vortex at the last stage was higher than that encountered in the lower-Reynolds-number tests.

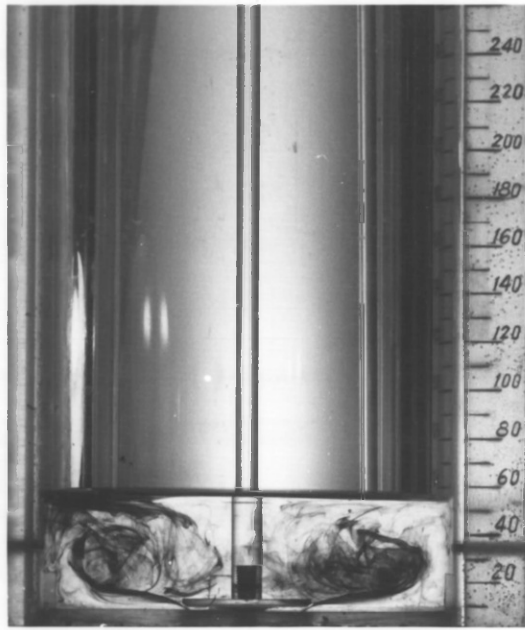
Figures 7.5-4 to 7.5-7 show four series of flow patterns for the same Reynolds number and valve geometry but for different values of the valve lift (i.e. tests No. 5 and No. 8 to 11). It was observed that:

- (1) at an early stage when the water level was low, the core of the main vortex was situated closer to the cylinder wall for a small lift than that for a larger lift;
- (2) the vortex shape for a small lift was less well defined because of

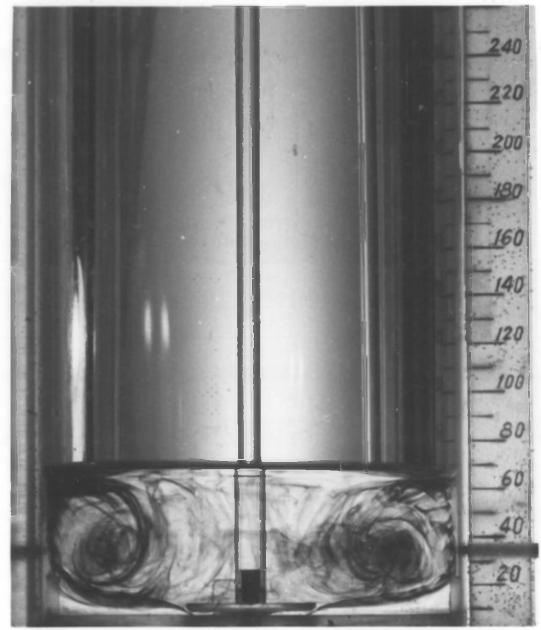
the higher flow velocity near the valve edge: and .

- (3) the angle between the cylinder axis and the streak lines through the valve opening was smaller for a larger valve lift as one might expect.

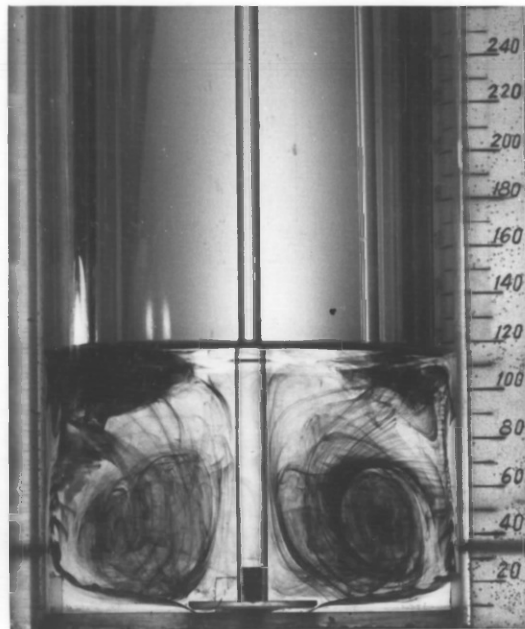
In Figure 7.5-8 are shown one series of flow patterns near a seatless valve for a Reynolds number of 620 and a dimensionless valve lift of 0.14 (i.e. Test No. 11 in Table 7.3-1), compared with the corresponding flow in Figure 7.5-1, for a valve with 45° seat angle. No significant differences can be noted at this lift test. However, it may be expected that at a smaller valve lift, the influence of the valve seat angle on the flow pattern is likely to be stronger.



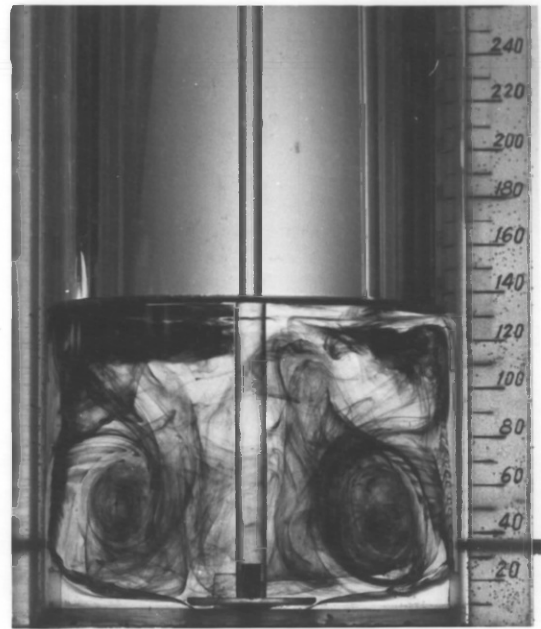
a. $t = 34$ seconds



b. $t = 64$ seconds

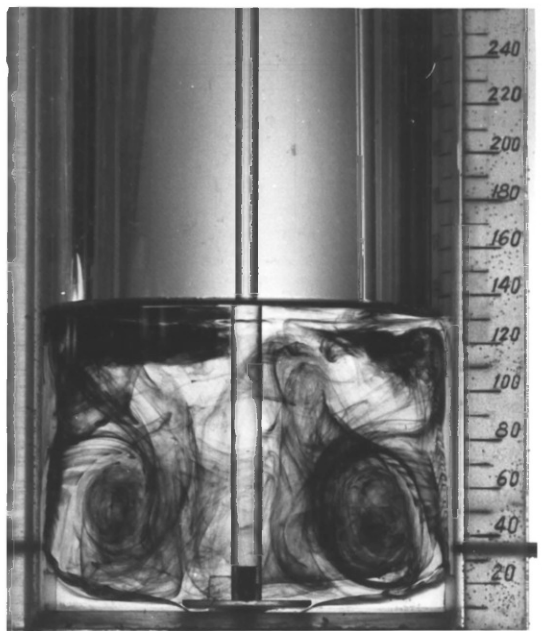
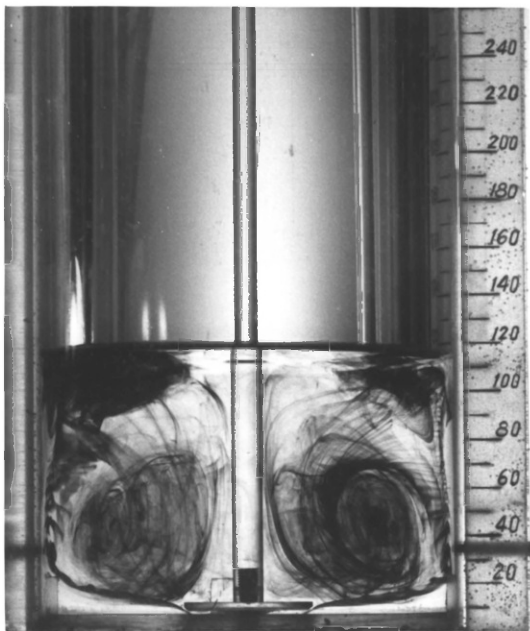
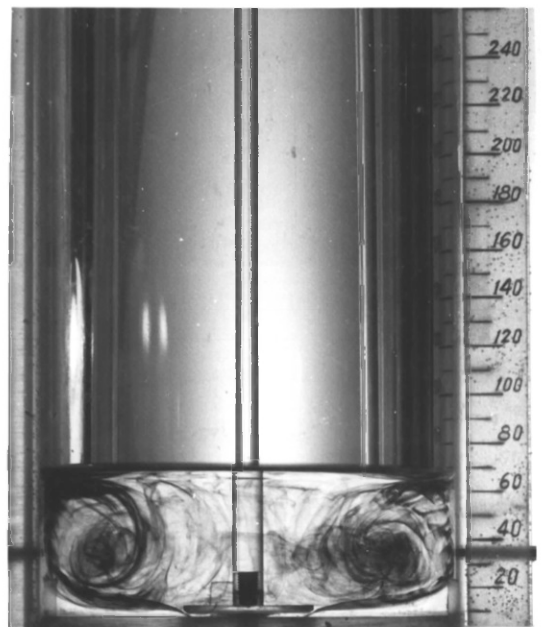
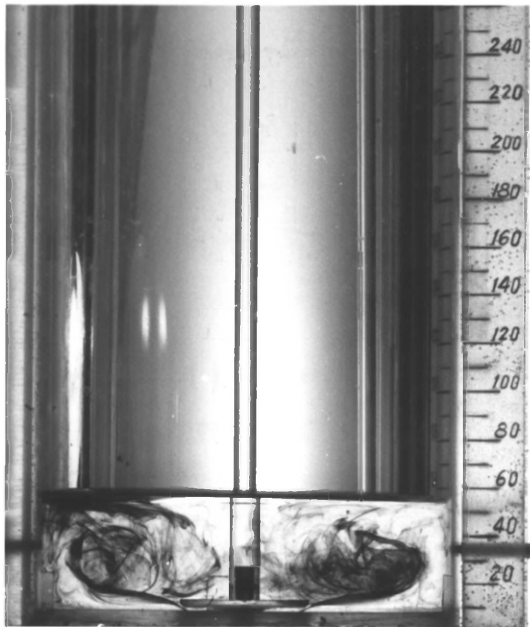


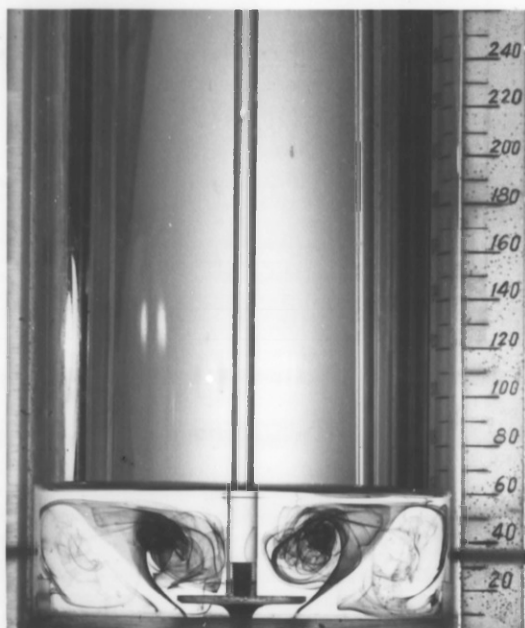
c. $t = 96$ seconds



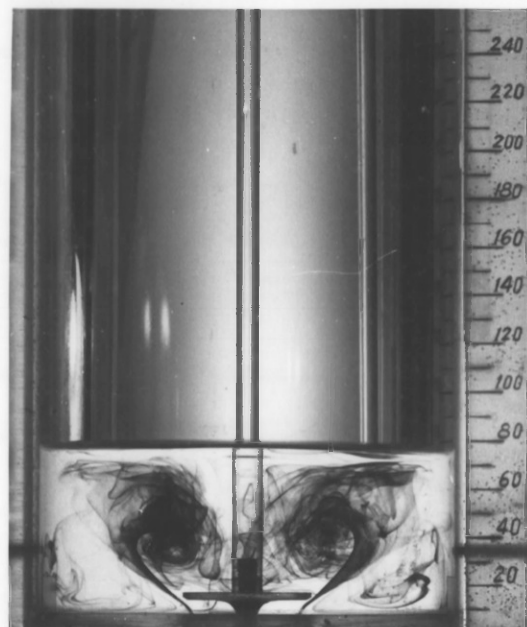
d. $t = 104$ seconds

Figure 7.5-4 Development of the flow pattern around a valve with seat angle of 45° ; $Re=620$, $L/D=0.08$.

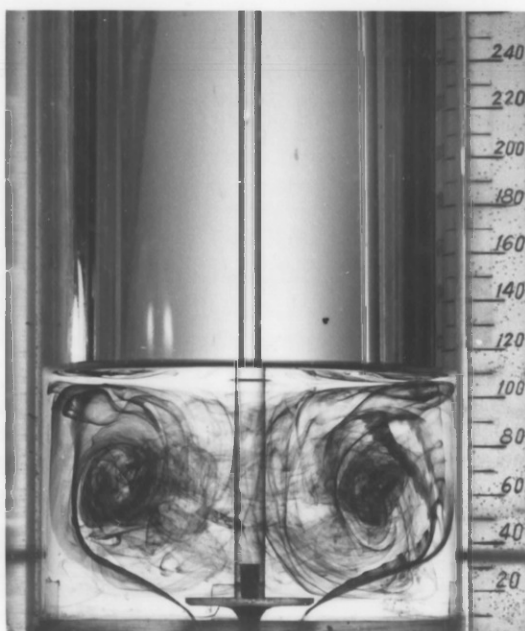




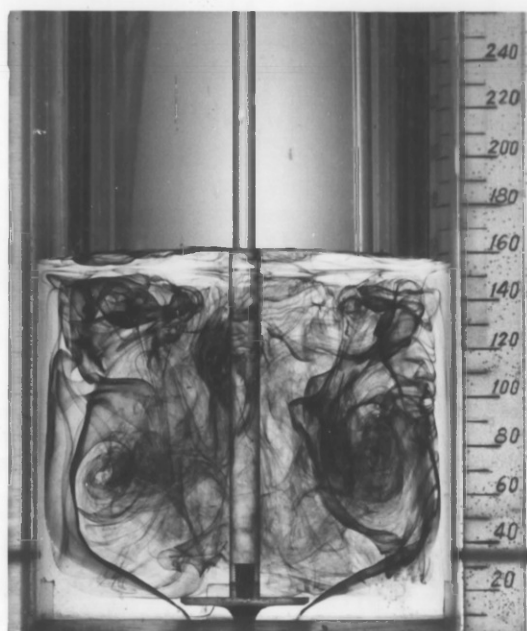
a. $t = 48$ seconds



b. $t = 62$ seconds

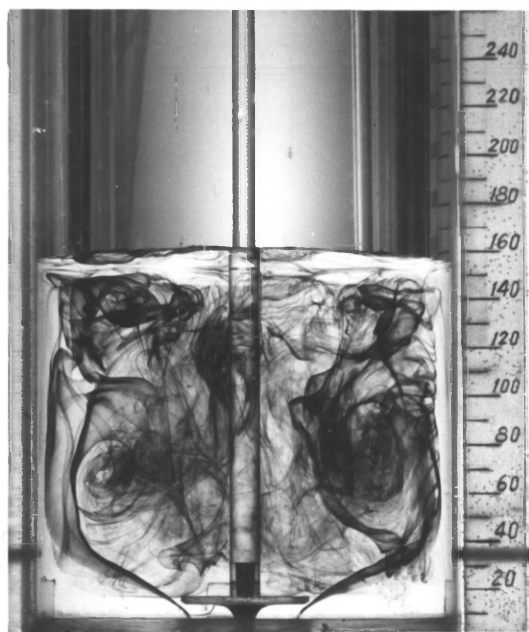
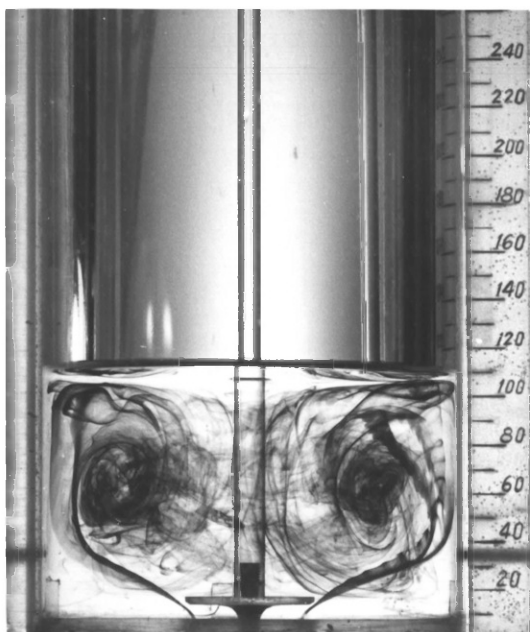
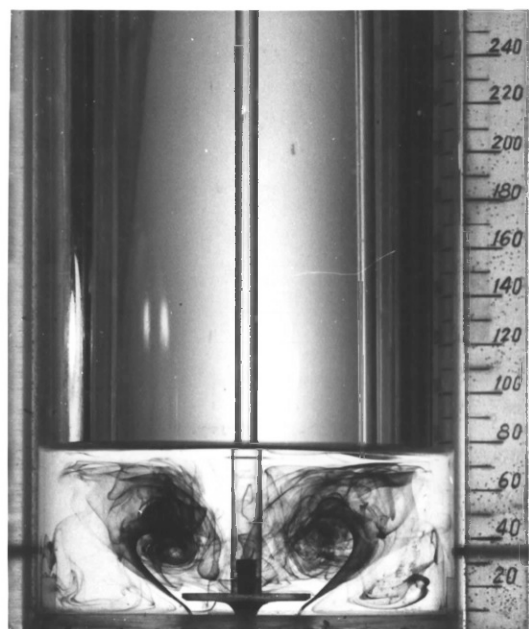
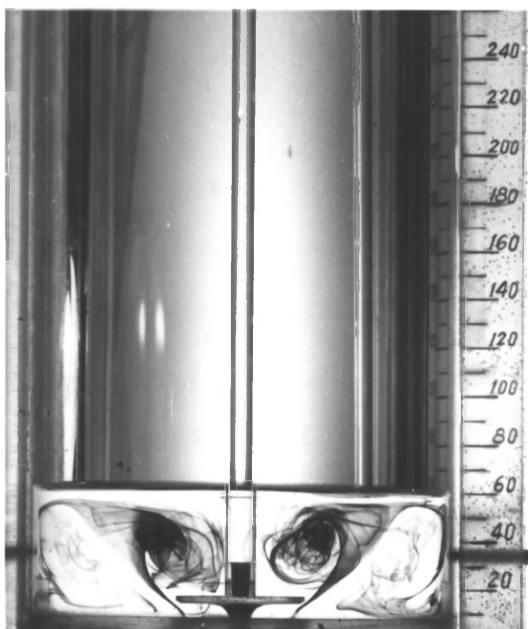


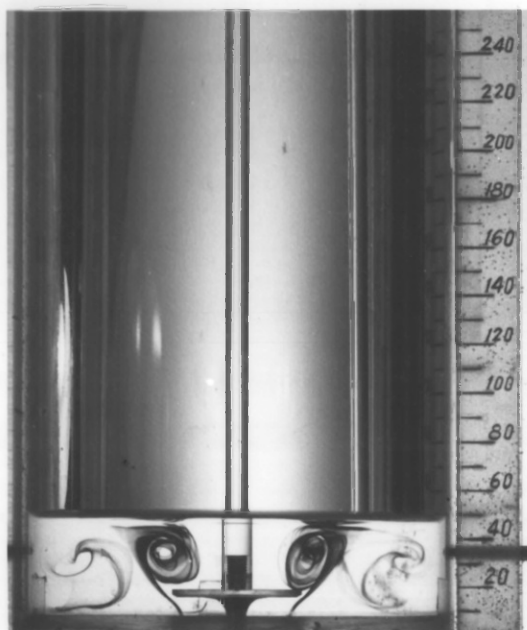
c. $t = 80$ seconds



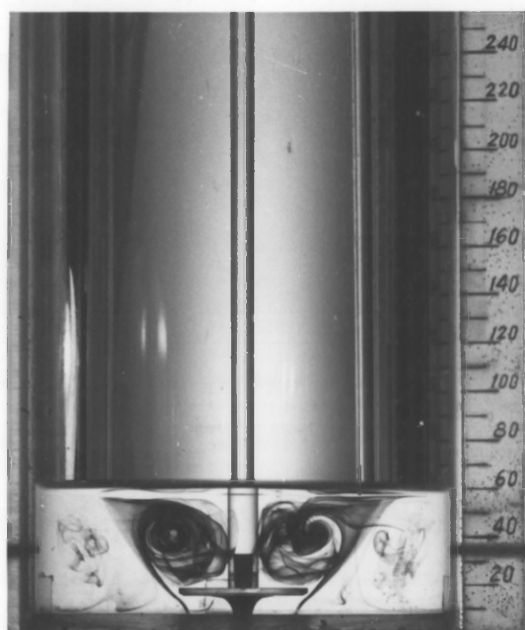
d. $t = 128$ seconds

Figure 7.5-5 Development of the flow pattern around a valve with seat angle of 45° ; $Re=620$, $L/D=0.14$.

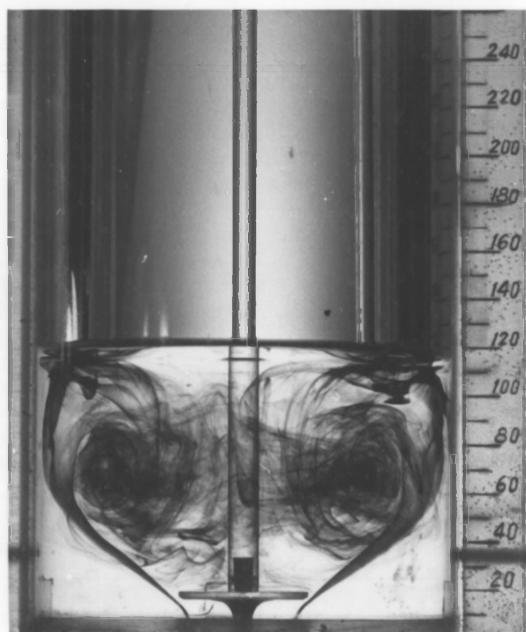




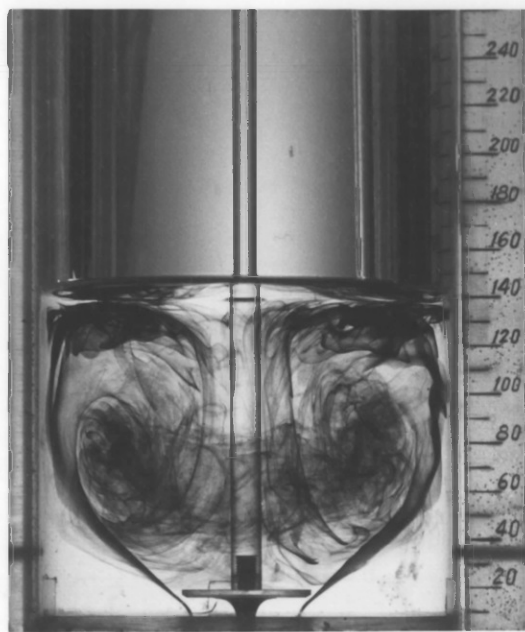
a. $t = 40$ seconds



b. $t = 48$ seconds

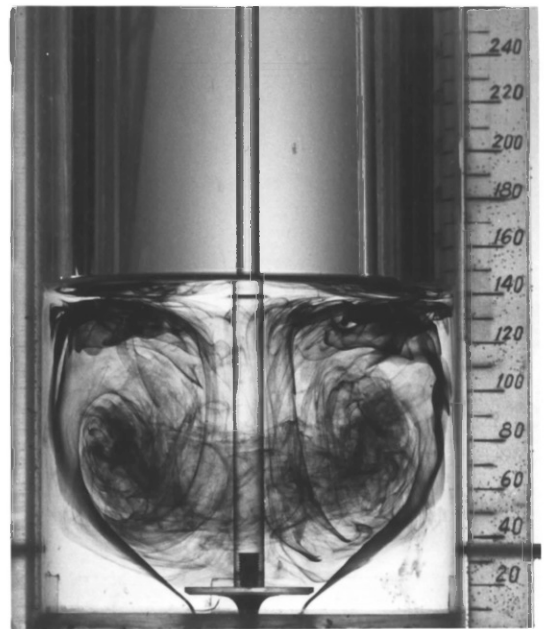
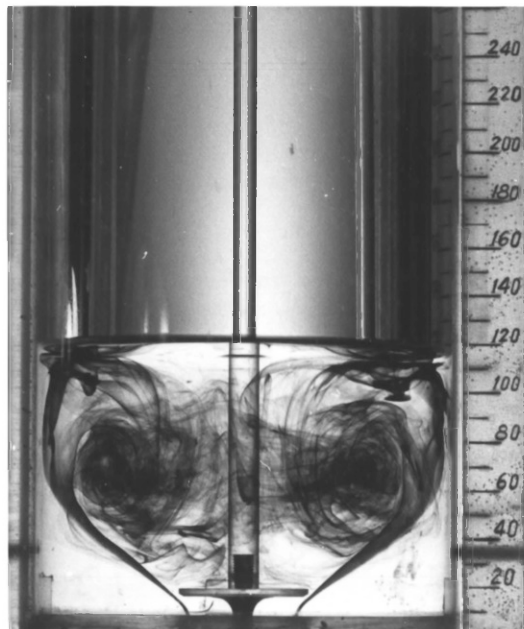
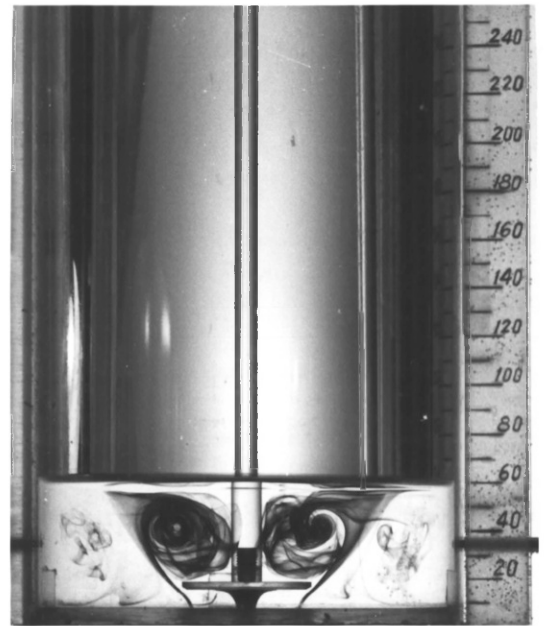
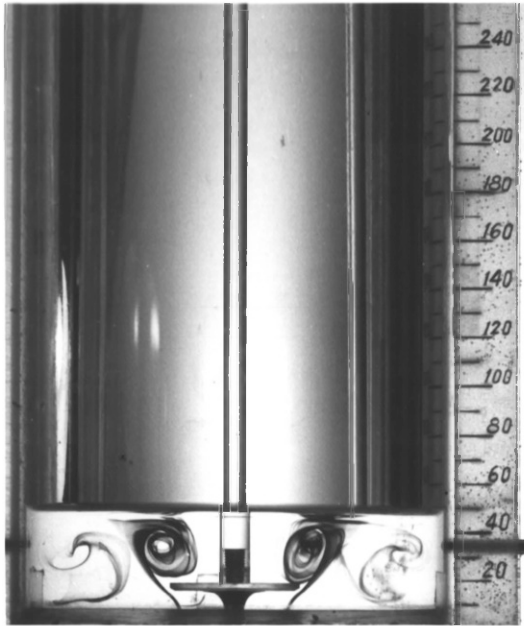


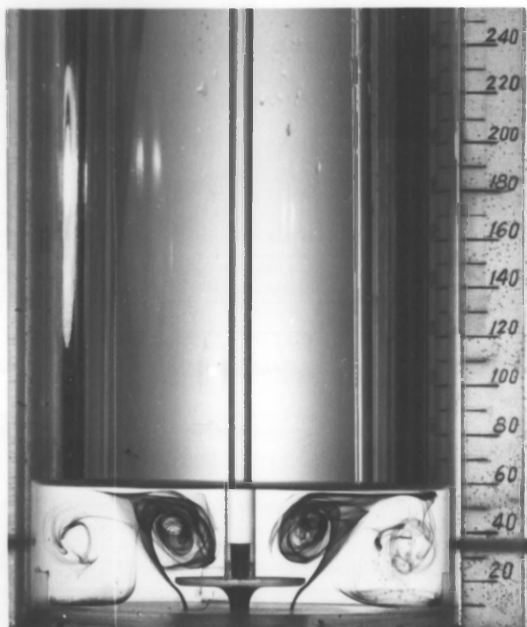
c. $t = 96$ seconds



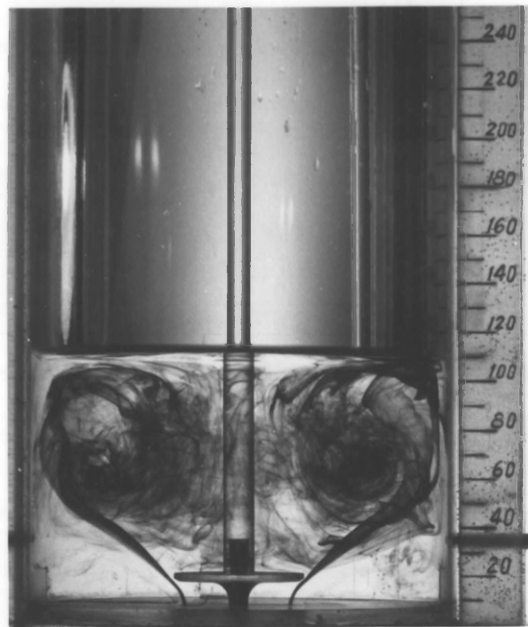
d. $t = 114$ seconds

Figure 7.5-6 Development of the flow pattern around a valve with seat angle of 45° ; $Re=620$, $L/D=0.2$.

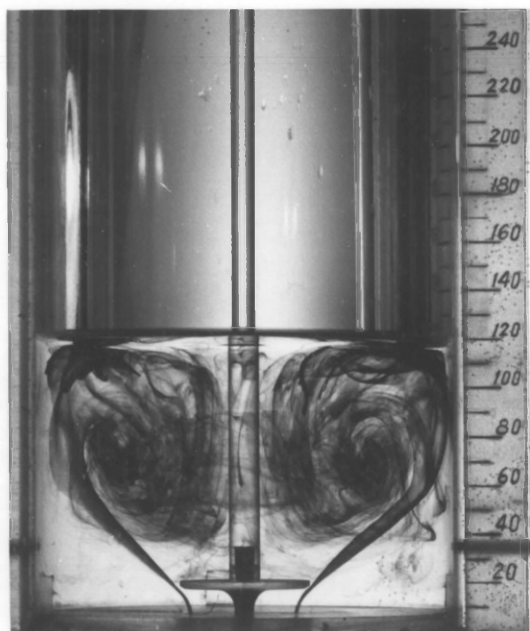




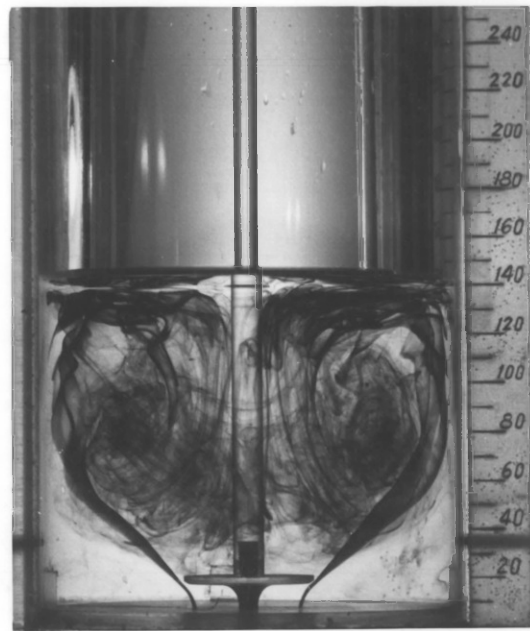
a. $t = 48$ seconds



b. $t = 88$ seconds

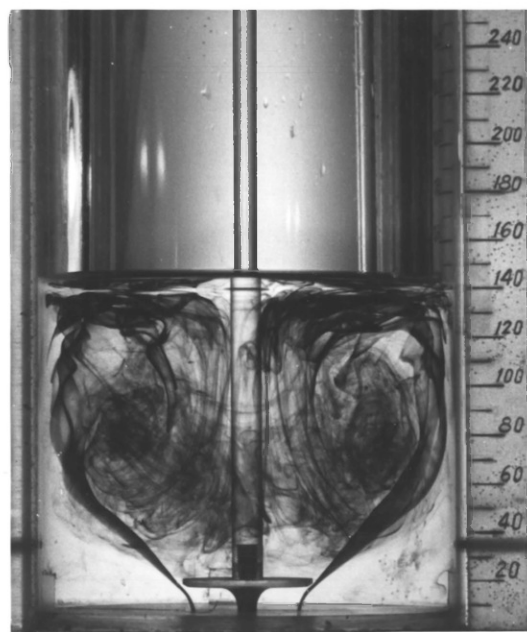
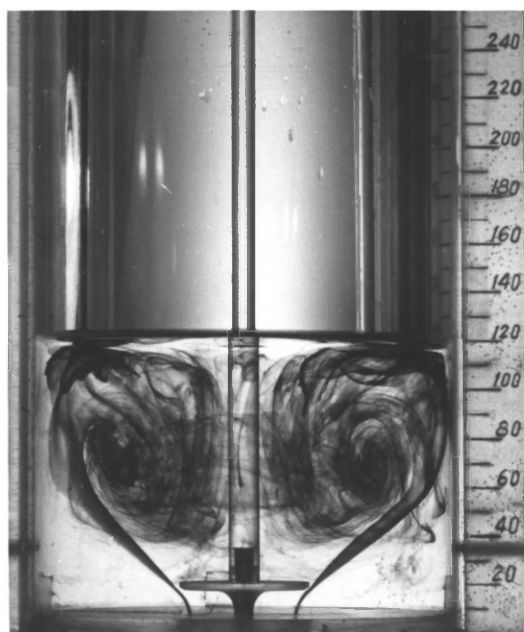
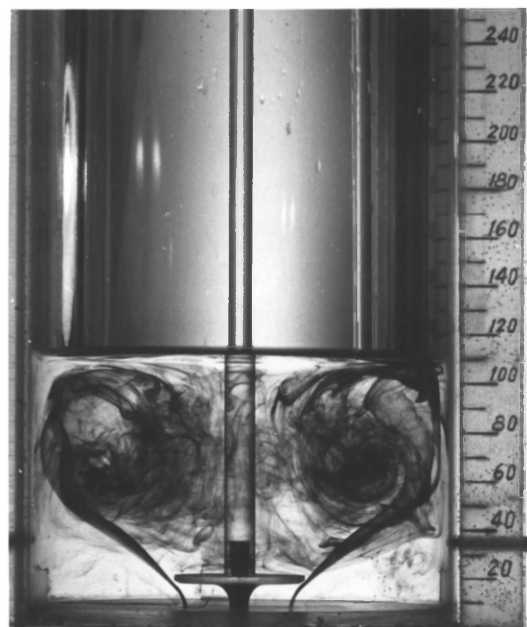
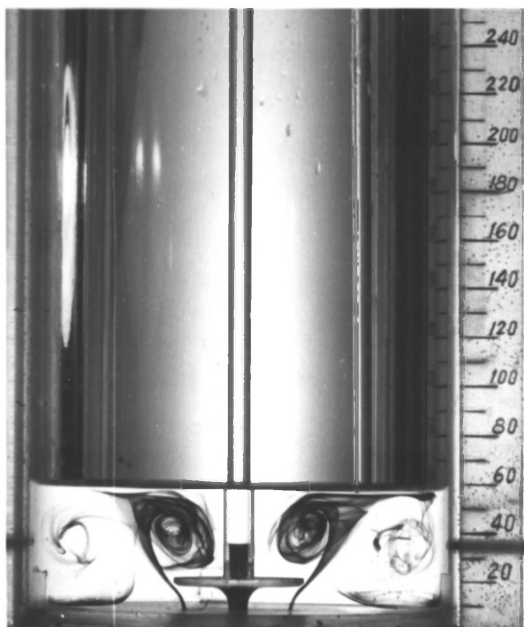


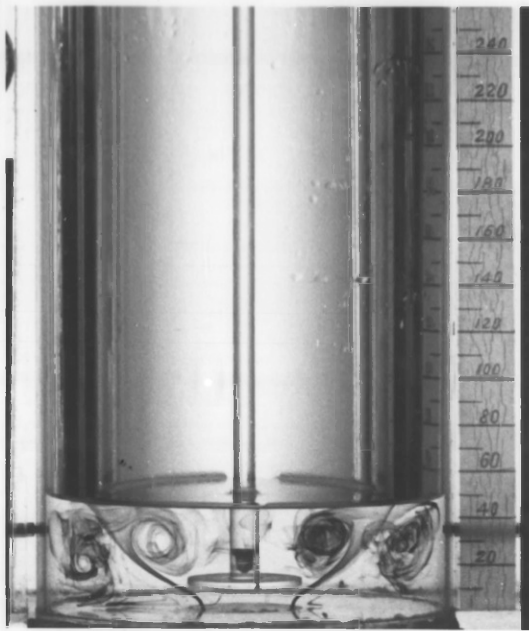
c. $t = 96$ seconds



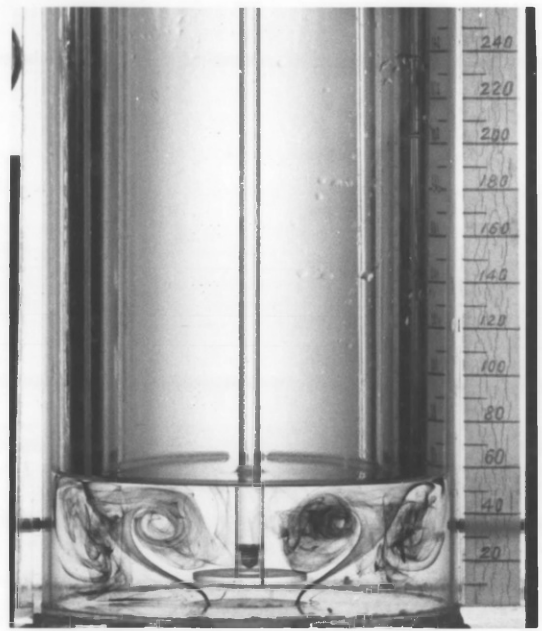
d. $t = 114$ seconds

Figure 7.5-7 Development of the flow pattern around a valve with seat angle of 45° ; $Re=620$, $L/D=0.24$.

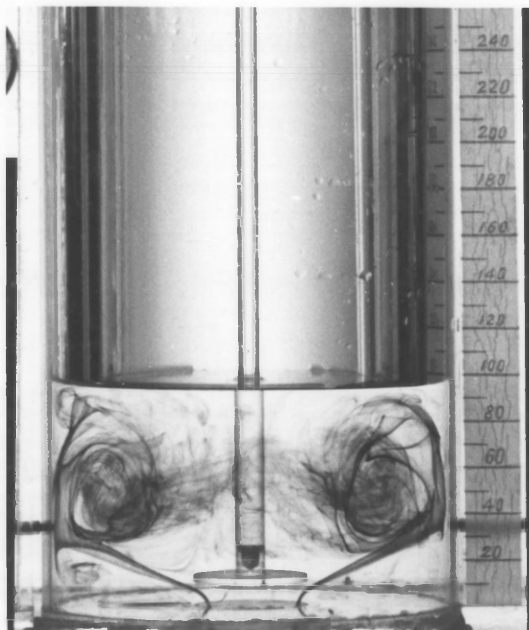




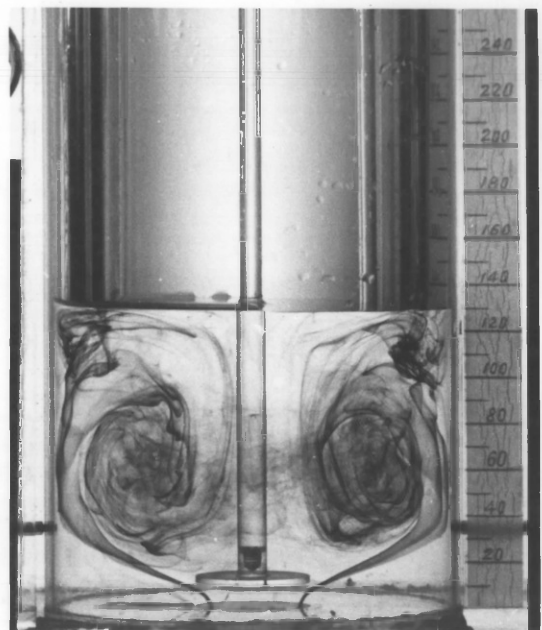
a. $t = 40$ seconds



b. $t = 50$ seconds

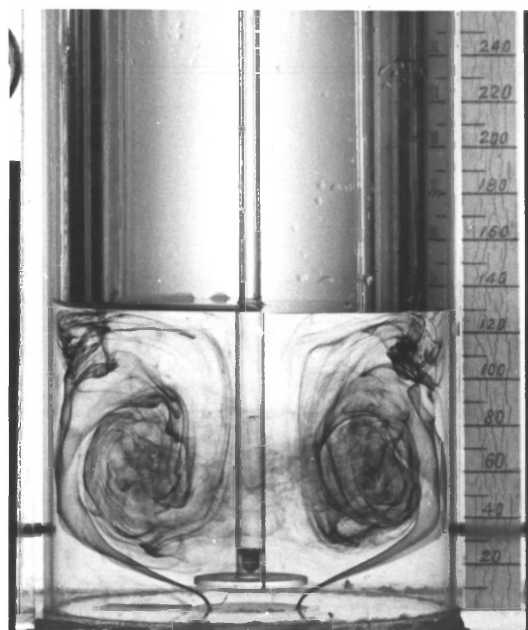
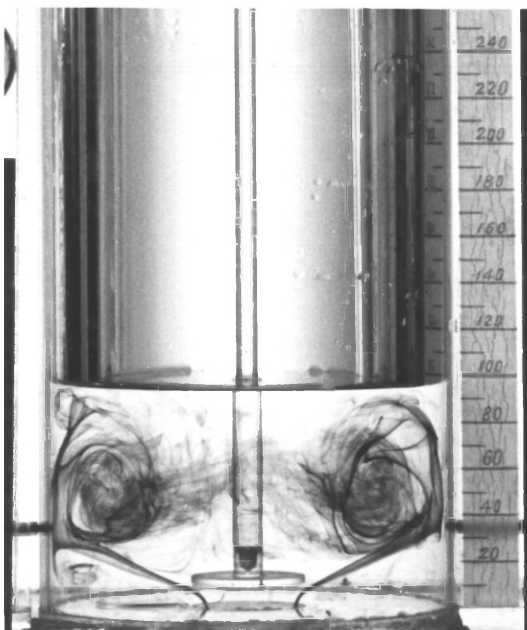
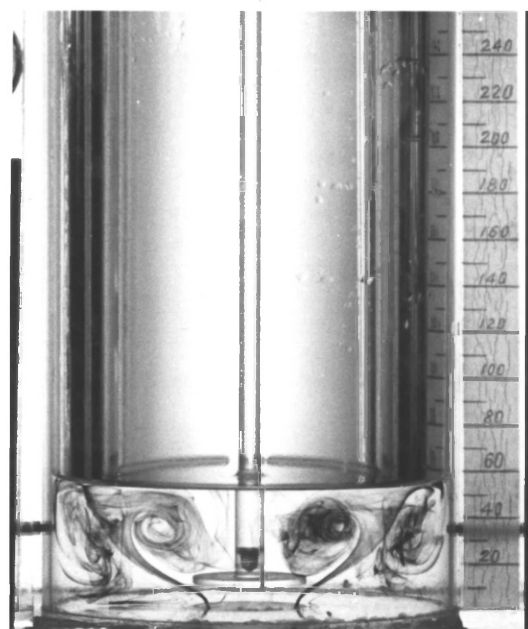
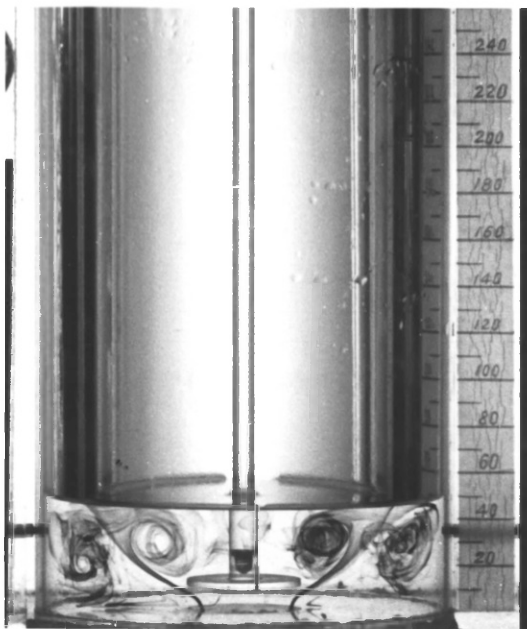


c. $t = 80$ seconds



d. $t = 104$ seconds

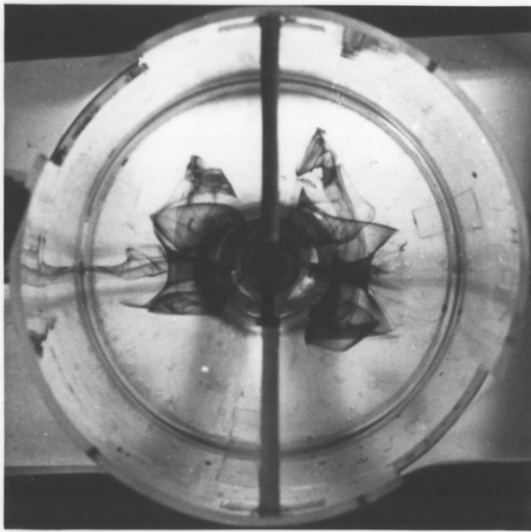
Figure 7.5-8 Development of the flow pattern around a seatless valve, $Re=620$, $L/D=0.14$.



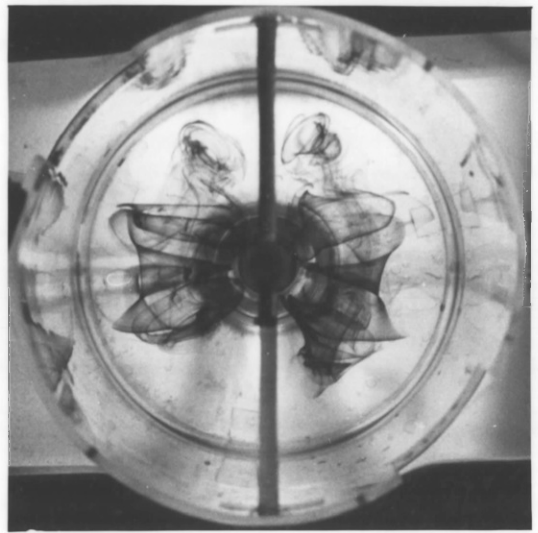
7.6 An Unexpected Phenomenon

It may be recalled that the two hypodermic tubes used to introduce the dyed fluid were axisymmetrically placed a short distance below the inlet in the supply-pipe. At a comparatively small Reynolds number of about 400, based upon the supply-pipe velocity and diameter, the flow would be expected to be laminar, and because of the axisymmetric nature of the flow, it would not be expected to spread by more than perhaps a few degrees in the circumferential direction. In fact, an unexpectedly large circumferential spread of the dye was observed. This can be seen from Figures 7.6-1 which show four sequences of the top view of the dye traces of the same flow as the side views of which have been shown in Figure 7.6-2. Figure 7.6-2 is a sketch emphasising the main features of the photographs: It is clear that the dye was not confined at, or in the vicinity of the plane through the axis of the cylinder and the two hypodermic tubes. The spiral line indicates an unexpectedly strong circumferential movement of the dye away from that plane in both directions. The motion is primarily along toroidal surfaces, and there is little transfer to the core of the vortex.

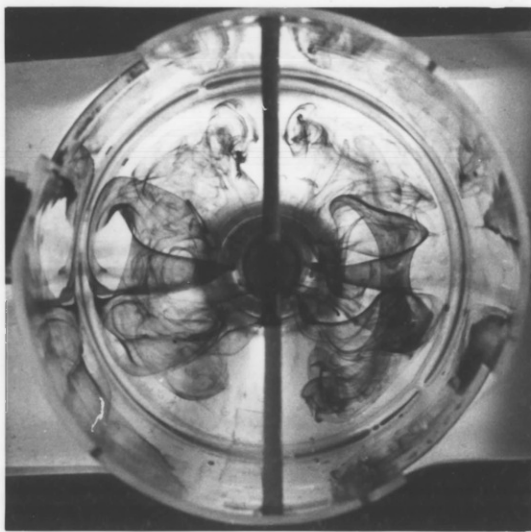
When the water surface reached a height of about 50mm, Figure 7.6-1(a), noticeable circumferential movement was observed. The pattern, 8 seconds later, at a water level height of 60mm, is shown in Figure 7.6-1(b). When the water surface reached 100mm, the circumferential motion had formed an almost complete torus, Figure 7.6-1(c). In Figure 7.6-1(d) is shown the pattern when the water surface had reached 120mm; the torus of dye traces had now extended outwards to the cylinder wall. Further increase in the water level brought more dye into this torus, forcing it against the cylinder wall; during



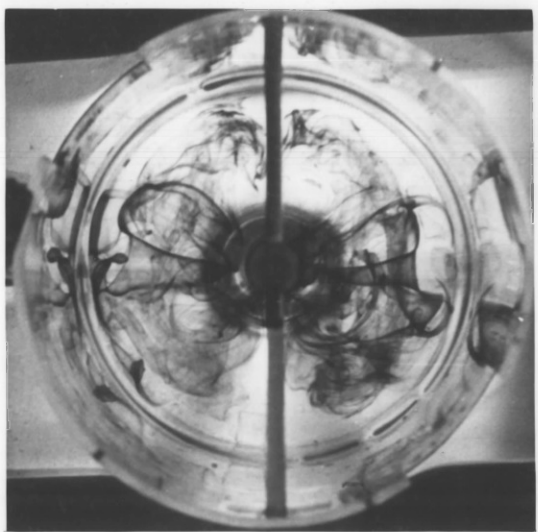
a. $t = 40$ seconds



b. $t = 48$ seconds

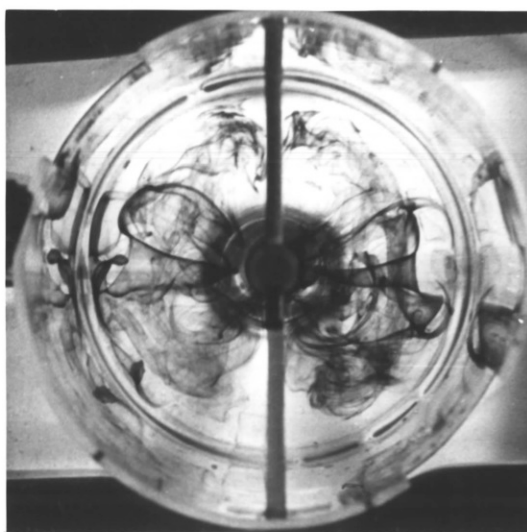
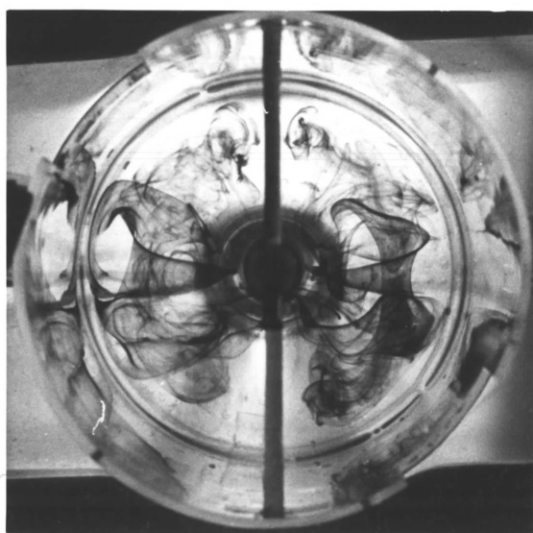
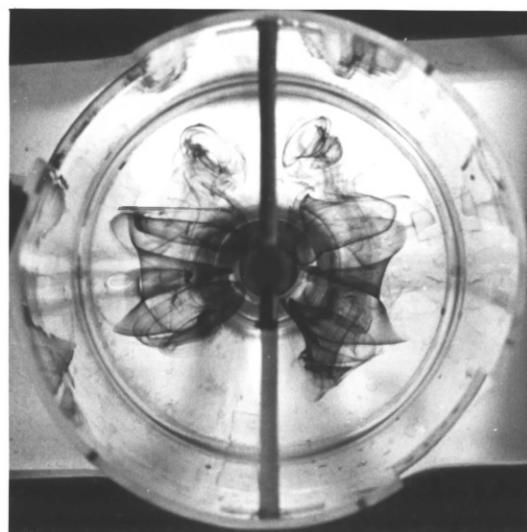
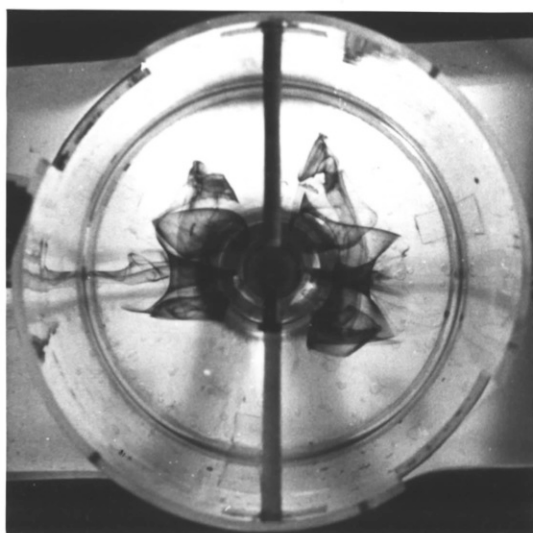


c. $t = 64$ seconds



d. $t = 112$ seconds

Figure 7.6-1 Top view of the flow pattern showing large circumferential movement.



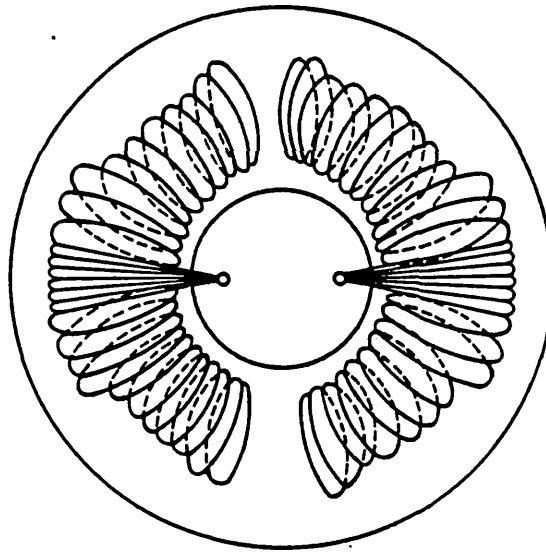
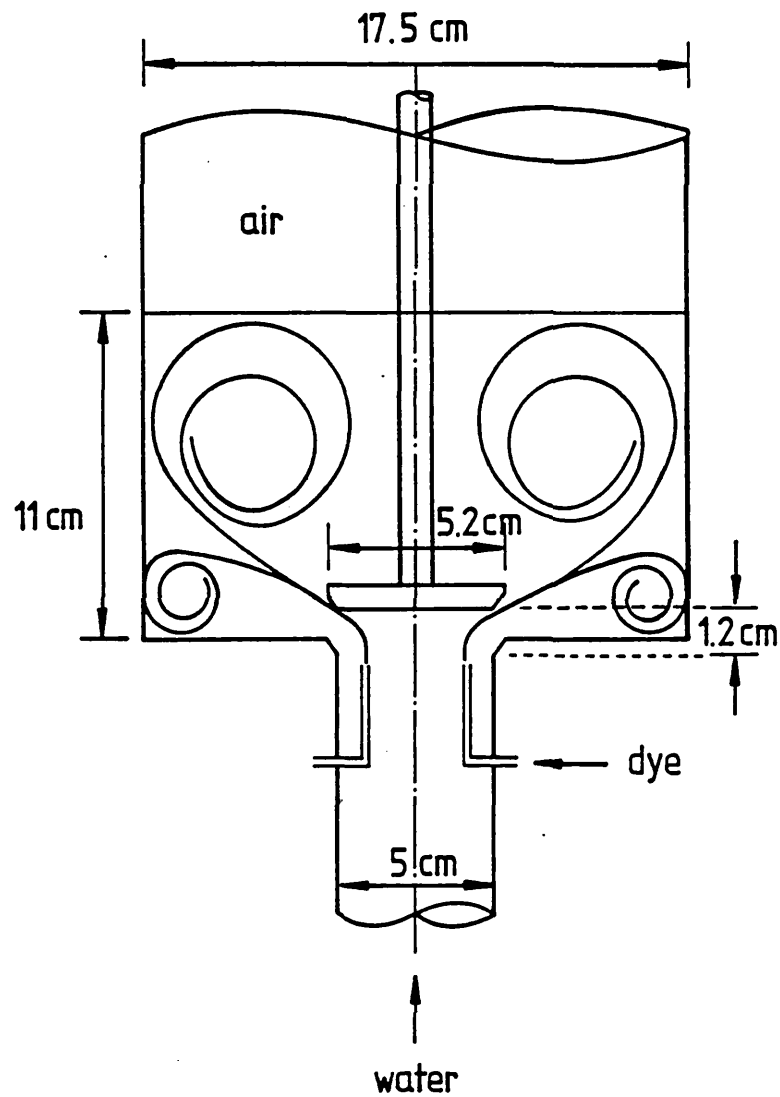


Figure 7.6-2 Sketch showing the circumferential movement of the dye.



this time more circumferential movement of the dye was observed.

This phenomenon was repeatedly observed throughout all the tests over the range of Reynolds number from 400 to 1000. Changing the dye-injection arrangement, e.g. with the two injection points at 90° instead of 180° to each other, as shown in the sketch of Figure 7.6-3, did not affect the appearance of the phenomenon described.

The unexpected circumferential motion shown in Figure 7.6-1 behaves as though there existed a large diffusion coefficient in the circumferential direction. If the process was indeed diffusion controlled, the same phenomenon should also appear upstream of the vortex; in fact nothing similar was observed there. Thus, diffusion cannot be responsible for the observed circumferential motion. However, the following explanation is advanced as being most plausible.

The phenomenon is likened to a "Kelvin wave", (Pedlosky, J. 1979 [108]) travelling in the direction perpendicular to the gravity acceleration. Let us first examine the situation represented by Figure 7.6-4, which shows the front of the interface of two immiscible fluids with slightly different densities travelling with velocity C_w . Simple analysis leads to:

$$C_w = \left(g \frac{\Delta \rho}{\rho} h \right)^{1/2} \quad , (7.6-1)$$

where $\Delta \rho$ is the density difference between the two fluids,

h is the step height of the interface between the fluids,

g is the gravitational acceleration.

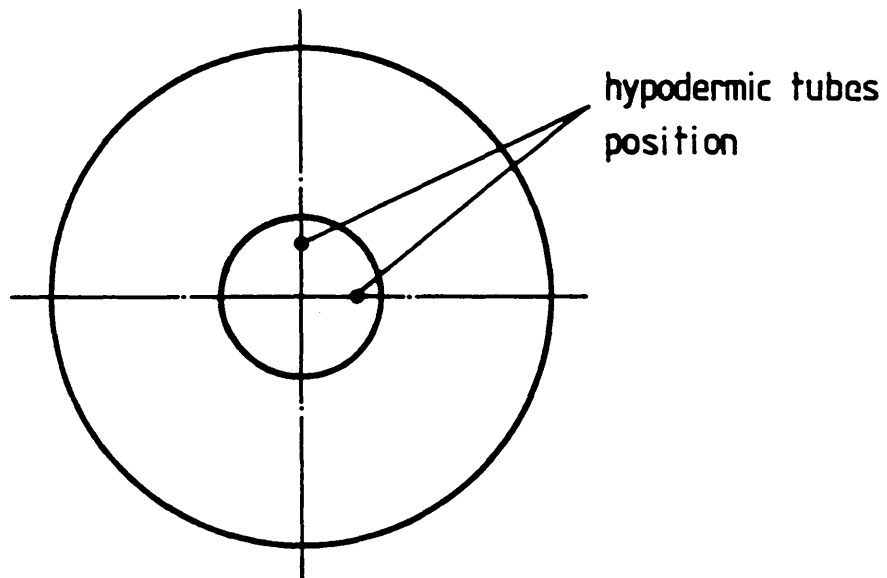


Figure 7.6-3 Sketch showing the position of the dye injection from the top view.

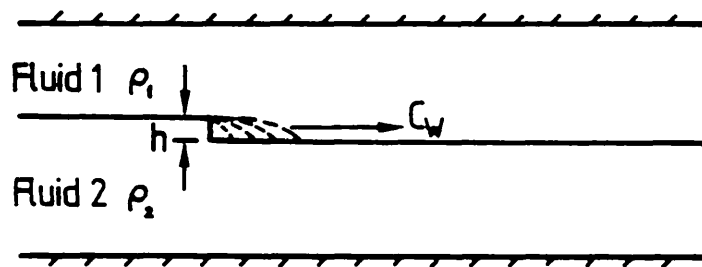


Figure 7.6-4 The inter-face line of a propagating Solitary Kelvin wave.

With reference to the observed phenomenon in the experiments, it is noted that the density of the dyed fluid differs slightly from that of the water. When a dyed-fluid "tube" reaches the vortex core downstream of the valve, it is subjected to a centrifugal acceleration towards the axis of the vortex. In a manner analogous to the Kelvin wave described above, waves carrying the dyed fluid travel in a direction perpendicular to the plane in which the centripetal acceleration acts, i.e. along the axis of the vortex. (See Figure 7.6-5). Because of the "double-fronted" situation, the dyed fluid spreads laterally along the axis of the vortex from both sides in opposite directions.

In accordance with equation 7.6-1, the speed of the dyed fluid is of the order of:

$$\left(\frac{\delta\rho}{\rho} gh\right)^{1/2}$$

where g is the centripetal acceleration, $g = \frac{w^2}{r}$.

w is the circumferential velocity about the vortex axis.

h is the thickness of the dyed fluid vortex.

r is the distance to the axis of the vortex.

So the speed of travel around the torous would be of the order of:

$$C_w = w \left(\frac{\delta\rho}{\rho} \frac{h}{r}\right)^{1/2} \quad 7.6-2$$

If $\frac{\delta\rho}{\rho}$ is assumed to be 0.005, and $\frac{h}{r}$ may be 0.05, the velocity of

the motion around the cylinder axis A-A is 0.016 of w .

From a typical record of the experiments, C_w was worked out as 1.31×10^{-4} m/s and w was established by averaging the field velocity

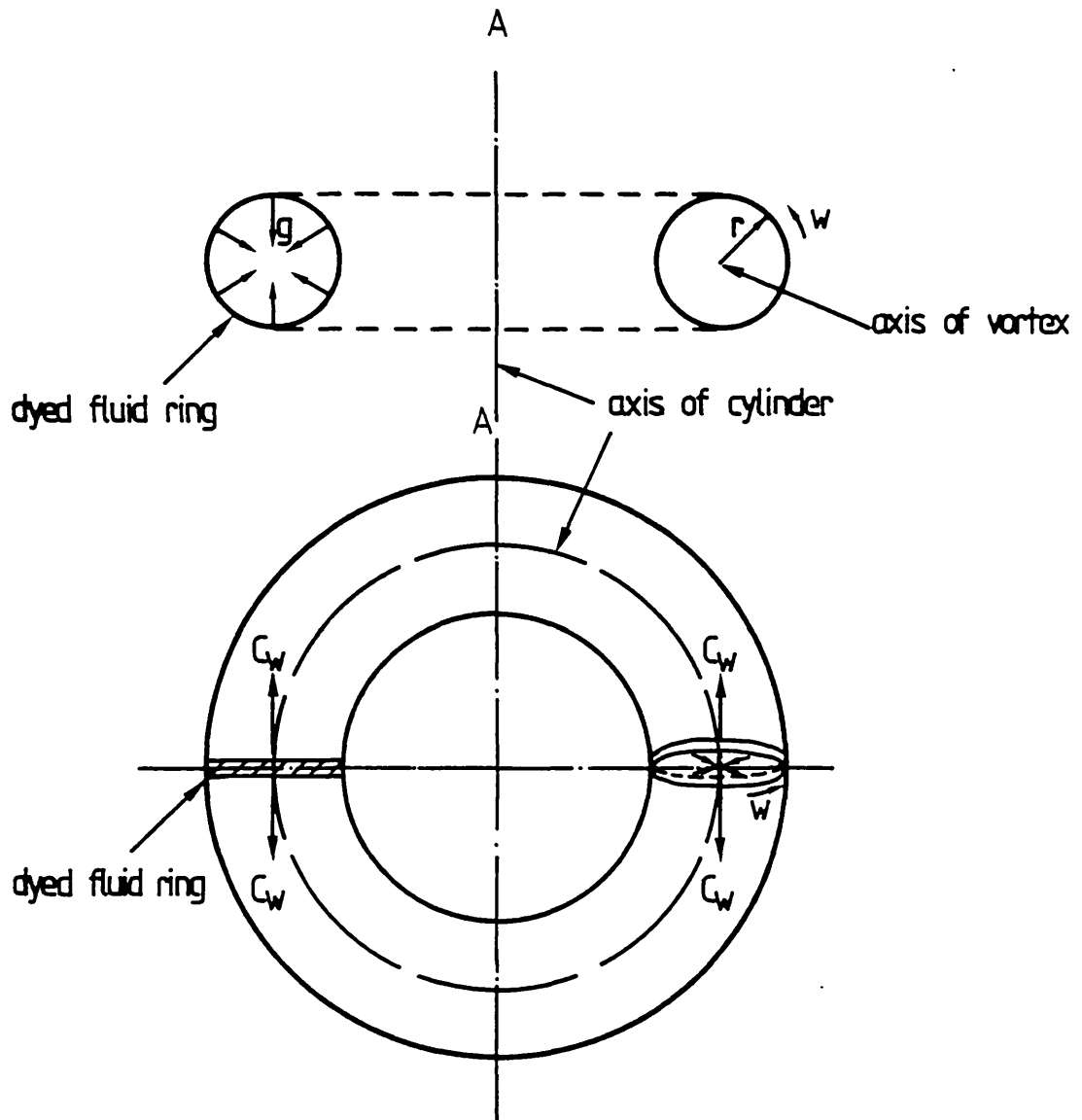


Figure 7.6-5 Sketch of the explanation of the circumferential motion.

values from the corresponding computer prediction to be $6.48 \times 10^{-3} \text{ m/s}$. Thus, C_w is 0.02 of w ; this accords, it appears, with the necessarily crude value obtained from equation 7.6-2. However, it must be emphasised that the above quantitative comparison should only be taken as a rough indication rather than as a conclusive proof of the validity of the explanation given.

7.7 Closure

A water-model rig has been constructed and used to observe the details of the laminar flow-patterns near an idealized-valve geometry, under both steady and unsteady flow conditions, and some parameters were studied. In doing so, it provides some quantitative results for computational simulations.

Some unexpected but apparently important features of circumferential motion have been revealed. A plausible explanation with a rough mathematical expression is advanced. However, it is desirable that more precise means of expressing the phenomena in mathematical terms should be derived. A further investigation will be needed, for toroidal-vortex flows are frequently encountered in combustion systems.

CHAPTER 8 - NUMERICAL SIMULATION OF THE EXPERIMENT

8.1 Introduction

In the experiments described in chapter 7, on flow of water through a coaxial valve-port, only simple measurements were made. Although flow patterns, as recorded on photographs, are also available, they can only yield gross features of the flow, such as the locations of the vortex centre and of the reattachment points. Quantitative information on the magnitude of the velocity at a particular point in the flow field could only be obtained with much more sophisticated equipment. However, this kind of quantitative information can now be revealed without much difficulty through numerical simulation of the flow, employing, for example, the same PHOENICS code as used for the computations described in chapter 4, 5 and 6.

The purpose of the present chapter is to describe the predictions made of the flows under the same geometric and flow conditions as those for several of the tests reported in chapter 7. Where possible the predictions are compared with the corresponding test results; but other quantitative information, such as velocity profiles, will also be presented to allow the characteristics of the flow to be examined.

Six sets of experimental conditions have been selected for the simulation exercise; these broadly cover the same range of the main parameters as in the tests. Details of the cases chosen are given in Table 8.1-1, in which both steady and unsteady flows have been included.

Table 8.1-1 Cases studied

state	test no. correspond- ing to chapter 7	valve geometry			Reynolds number Re
		seat angle θ	valve lift L_v/D_v	valve stem diameter $d_s(m)$	
steady	1	0°	0.14	0	400
	2	0°	0.14	0	620
	3	0°	0.14	0	1000
	4	45°	0.14	0.008	620
	11	0°	0.14	0	620
	5	45°	0.14	0.008	620

There it can be seen that the physical situation considered has low values of the Reynolds number, so all the flows are assumed to be laminar and axisymmetrical. The governing equations which need to be solved are thus the continuity and the two momentum equations expressed in polar coordinates. As recirculation within the flow domain is expected, the fully elliptic form of the differential equations must be used.

The remainder of this chapter comprises three sections. In section 8.2, details of the predictions of the steady flows are given and the predicted streamline patterns are compared with those of the corresponding experiments. Section 8.3 is concerned with the predictions of the unsteady flows. For these, use was made of a moving-grid system and a particle-tracking technique for tracking the dye trajectories; details of these will be described, along with a comparison with the corresponding experimental results. Concluding remarks on the present series of

numerical simulations are contained in section 8.4.

8.2 Steady-Flow Computations

8.21 The Phenomena to be Predicted

Water flows steadily from a vertical pipe of small diameter into a coaxial cylinder of a large diameter through the partial obstruction of a valve. Two regions of recirculating flow appear immediately downstream of the enlargement section and the valve (see Figure 8.21-1). This flow exhibits some complex features. It is, however, desirable to be able to calculate such flow and compare with the experimental measurement. In chapters 5 and 6, the comparison between prediction and available experimental data was only for discharge coefficient; in the present study, however, the length of the recirculation and the position of the vortex core are compared with the experiment as well, and the distribution of velocity is also predicted for examining the flow.

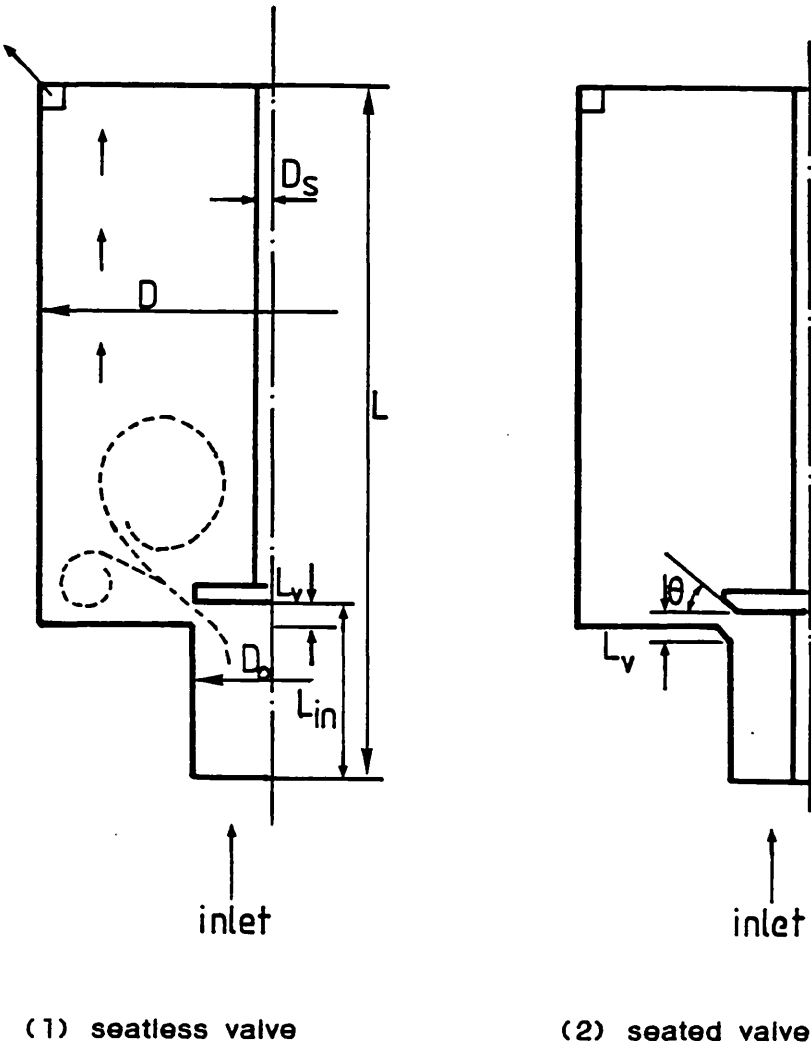


Figure 8.21-1 Flow through a coaxial valve port geometry and dimensions.

8.22 Geometry and Dimensions

The flow configuration considered is shown in Figure 8.21-1 and details of the dimensions are given in Table 8.22-1. The inlet plane is located 0.056m upstream of the step wall, i.e. cylinder head, and the slipwall boundary of the domain is the water-air interface, situated at 0.346m downstream of the step wall.

Table 8.22-1 Dimensions of the valve-port

	seatless valve	valve with 45° angle
Inlet Length L_{in}	0.056m	0.056m
test section Length L	0.4m	0.4m
Inlet pipe diameter D_0	0.05m	0.05m
cylinder diameter D	0.179m	0.179m
valve diameter D_v	0.05m	0.052m
valve thickness W_t	0.007m	0.007m
valve seat angle θ	0	45°
dimensionless valve lift L_v/D_v	0.14	0.14
valve support diameter D_s	0.008m	0.008m

8.23 Flow-Domain for Computation and Grid-Distribution

Table 8.23-1 presents a summary of the different cases studied and grids used. In each case, the grid is distributed non-uniformly in both axial and lateral directions.

Table 8.23-1 Steady cases studied.

case no corresponding to Chapter 7	Re Reynolds No	Valve situation			Grid number NY x NZ
		angle θ°	lift L/D_v	stem D_s	
1	400	0	0.14	0	27 x 45
2	620	0	0.14	0	27 x 45
3	1000	0	0.14	0	27 x 45
4	620	45	0.14	0.008	27 x 48

The calculation domain is shown in Figure 8.23-1; also shown there is a typical layout of the grid network used. The grid lines are distributed non-uniformly over the flow domain with a larger number of nodes near the valve and the cylinder wall regions. A total of 27 grid-nodes are used to span a radius of 0.0865 m out of which 10 uniformly-spaced nodes are used to cover the inlet duct of 0.025 m radius as shown, the rest is non-uniformly distributed for the step. A total of 45 axial nodes are used for the seatless valve, and 48 for the valve with a 45° angle.

Grid independence for a similar problem (chapters 5 and 6) has already been established for this type of calculation domain and the same grid is expected to be adequate for the present calculations. In order to make sure that grid independence is achieved, four different grid systems have been investigated; these are cited in Table 8.23-2, along with the predicted values of the velocity components for case 1 at a chosen point A, (in the important region, i.e. between the step wall and the upstream face of the valve, ($Y_A/D = 0.253$, $Z_A/D = 0.357$) Figure 8.23-1). It can be seen that the velocity values do not significantly change beyond the 27x45 grid nodes.

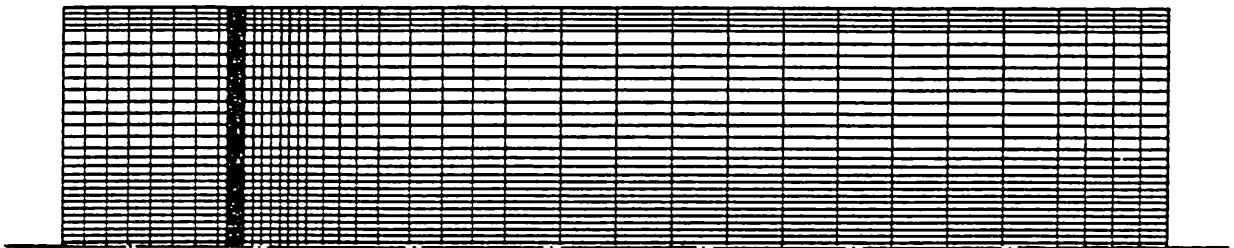


Figure 8.23-1. The calculation domain and a typical grid distribution (NY x NZ: 27 x 45).

Table 8.23-2. Effect of grid nodes on the velocity value at A ($Y_A/D = 0.253$, $Z_A/D = 0.357$).

grid number	$W_A/\bar{W}_{inlet}$	$V_A/\bar{W}_{inlet}$
25 x 35	2.644 E-2	1.798 E-1
27 x 39	2.611 E-2	1.776 E-1
27 x 45	2.591 E-2	1.769 E-1
27 x 48	2.588 E-2	1.767 E-1

8.24 Boundary conditions

The boundaries of the computational domain comprise the symmetry axis, the inlet plane and outlet boundary and the walls, as shown in Figure 8.24-1.

Axis of symmetry

Because of symmetry, it is appropriate to specify $V = 0$ and $\partial W/\partial y = 0$ on this boundary.

Inlet plane

For the present calculations the following conditions were imposed at the inlet:

$$V = 0;$$

parabolic distributions of the axial velocity across the inlet duct were

specified, as shown in Figure 8.24-1 for valves with and without a stem. For a valve without a stem formula 8.24-1 was used; and for a valve with a stem formula 8.24-2.

$$W_z = \bar{W} \left[1 - \left(\frac{y}{R} \right)^2 \right] \quad , (8.24-1)$$

$$W_z = \bar{W} \left[(R_1^2 - y^2) + \frac{(n^2-1) R_1^2}{\ln n} \ln \frac{y}{R_1} \right] \quad , (8.24-2)$$

where W_z is the axial velocity;

$\bar{W}$ is the average axial-velocity at the inlet;

$n = \frac{R_2}{R_1}$ is the ratio of two radii R_1 and R_2 .

Slip Wall

The water surface is treated as a slip wall, where the convection and diffusion fluxes were zero. These conditions were introduced by setting normal velocity and normal gradient of velocity to zero, i.e. $W=0$ and $\frac{\partial v}{\partial z} = 0$.

Outlet holes

For the outlet holes situated at the cylinder wall (the same level as the water surface), a constant pressure condition was imposed, the mass flow cross the boundary being calculated from continuity.

Wall boundary

All walls (including the cylinder wall and head, the valve surfaces and the valve support) were stationary and impervious; both v and w were set to zero on them.

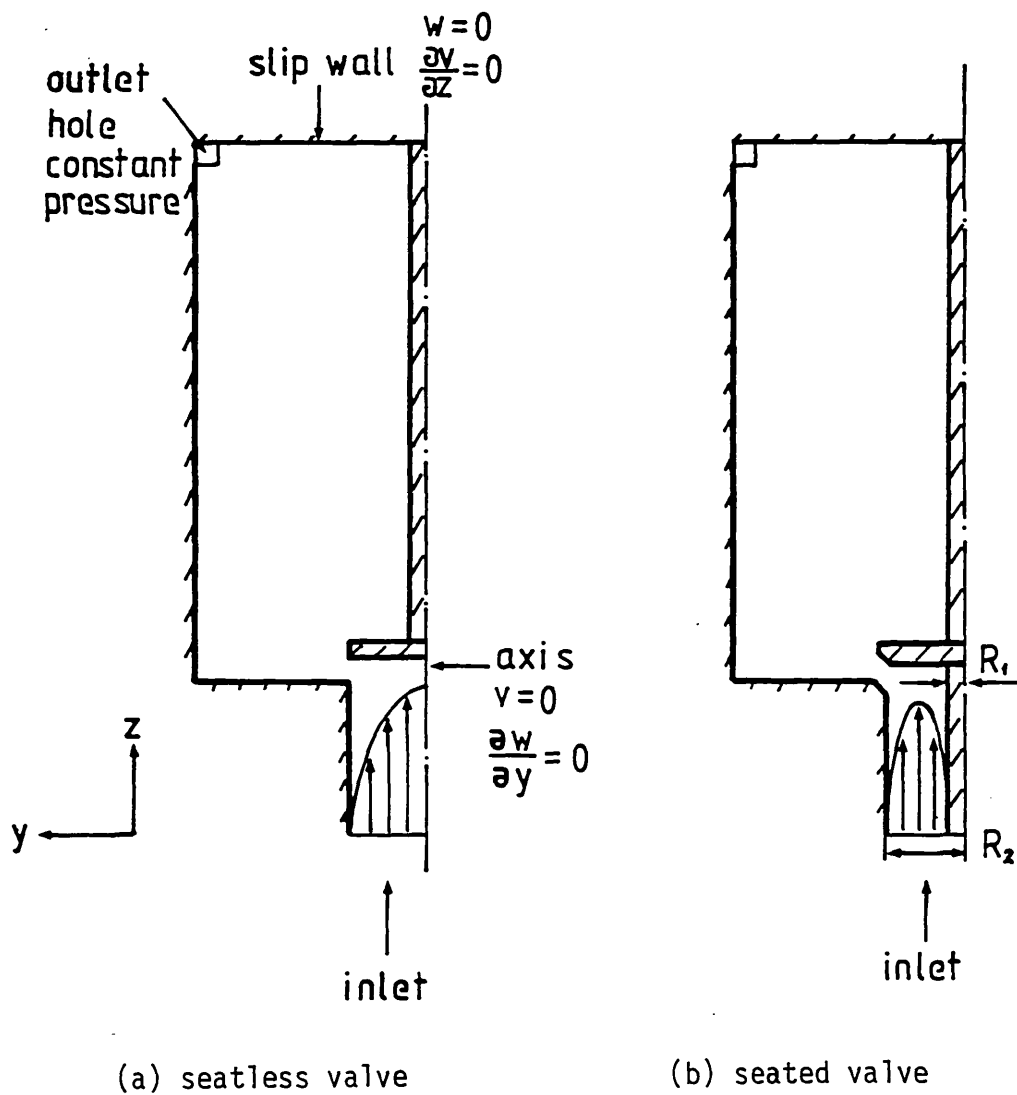


Figure 8.24-1. The boundaries of flow domain.

8.25 Results and Discussion

The Length of Recirculation Zone

Table 8.25-1 shows the predicted and measured values of the length of the recirculation zone for the corner vortex normalized by the diameter of the valve, L_R/D_v . L_R is defined in the present computation as the axial length of the first reverse-flow region, which is between the step and the cylinder wall (i.e. to the position where the mean streamline is reanchored to the cylinder wall).

Table 8.25-1 Measured and predicted the length of the recirculation zone

	Re	L_R/D_v	L_R/D_v	seat angle θ
Case No	Reynolds number	Experiment data	Present prediction	
1	400	0.35	0.36	0°
2	620	0.36	0.374	0°
3	1000	0.374	0.378	0°
4	620	0.365	0.371	45°

The maximum discrepancy between the predicted and measured values shown in Table 8.25-1 is 4%. It may be considered that the prediction is acceptable considering the measurement error of L_R .

Distribution of Velocity and Streamlines

In Figures 8.25-1 to 8.25-3 are shown the predicted axial-velocity distributions at four selected locations for three Reynolds numbers, based

upon the diameter of inlet duct, of 400, 620 and 1000 respectively (i.e. cases 1 to 3 of Table 8.1-1 for seatless valve); while Figure 8.25-4 shows normalized axial-velocity distribution at the same four locations for a valve with 45° seat angle, the Reynolds number is 620. The velocity at each section is normalized with the mean velocity (W_{in}) of 0.008 m/s, 0.0124 m/s and 0.02 m/s for flows of three different Reynolds numbers.

For the first three Figures, very similar parabolic profiles are seen near the entrance, at $Z/D_v = 0.08$, profile 1. Profile 2 at $Z/D_v = 1.188$, i.e. downstream of the step but upstream of the valve, changes sign near the cylinder wall, indicating recirculation there, caused by the sudden enlargement, and remains the maximum value near the axis. Profiles 3, at $Z/D_v = 1.764$, i.e. at a section downstream of the valve, indicate the presence of another recirculation, just downstream of the valve. Profiles 4, at $Z/D_v = 7.694$, i.e. further downstream of the valve and near the exit boundary, reveal that the axial velocity at that plane is larger near the cylinder wall where exit holes are.

It can also be seen from Figure 8.25-1 to 3 that the larger the Reynolds number, the larger the size of the recirculation zone downstream of the valve.

Figure 8.25-4 shows velocity distribution for a valve with a 45° angle, because of the stem upstream of the valve, the velocity distribution at the inlet is assumed by a different shape; beyond the step all the profiles have the same characteristics as those discussed above for the seatless valve.

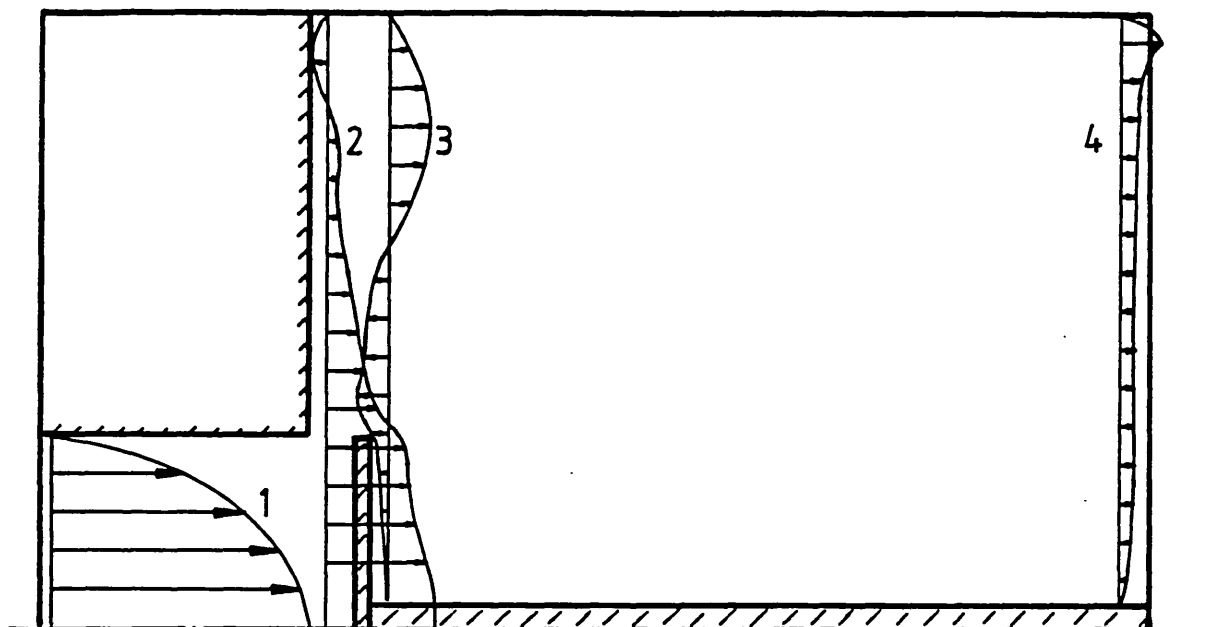


Figure 8.25-1 Axial velocity profiles at four locations:

(1) $Z/D_V = 0.08$; (2) $Z/D_V = 1.188$;

(3) $Z/D_V = 1.764$; (4) $Z/D_V = 7.694$;

$Re = 400$, $L_V/D_V = 0.14$, $\theta = 0^\circ$.

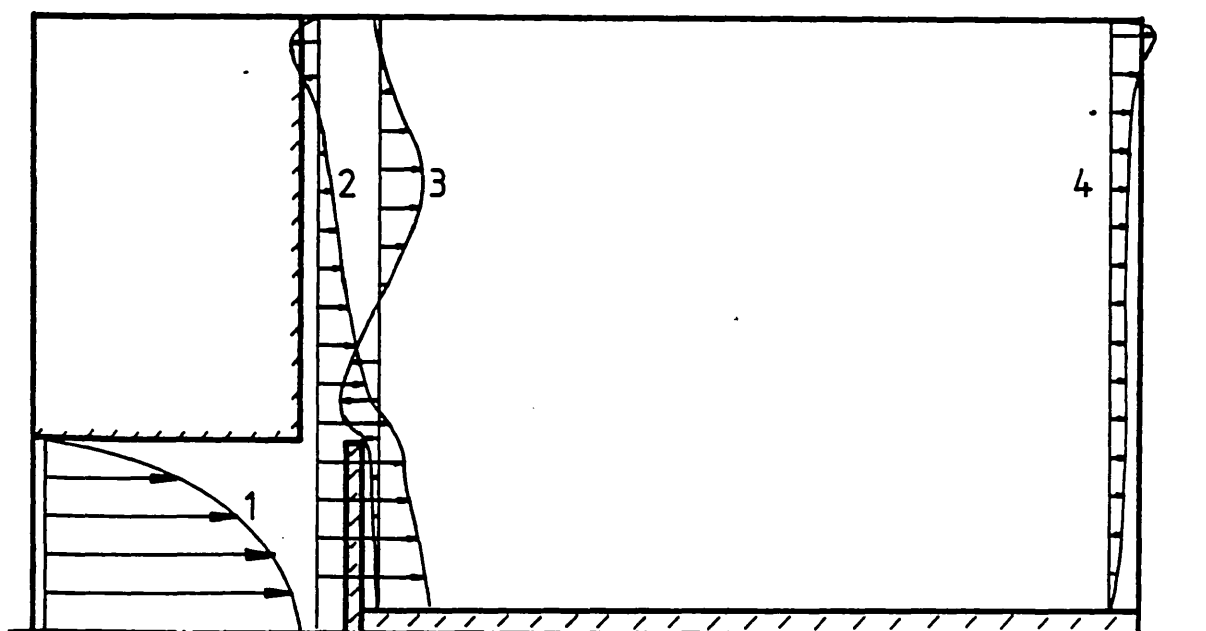


Figure 8.25-2 Axial velocity profiles at four locations:

(1) $Z/D_V = 0.08$; (2) $Z/D_V = 1.188$;

(3) $Z/D_V = 1.764$; (4) $Z/D_V = 7.694$;

$Re = 620$, $L_V/D_V = 0.14$, $\theta = 0^\circ$.

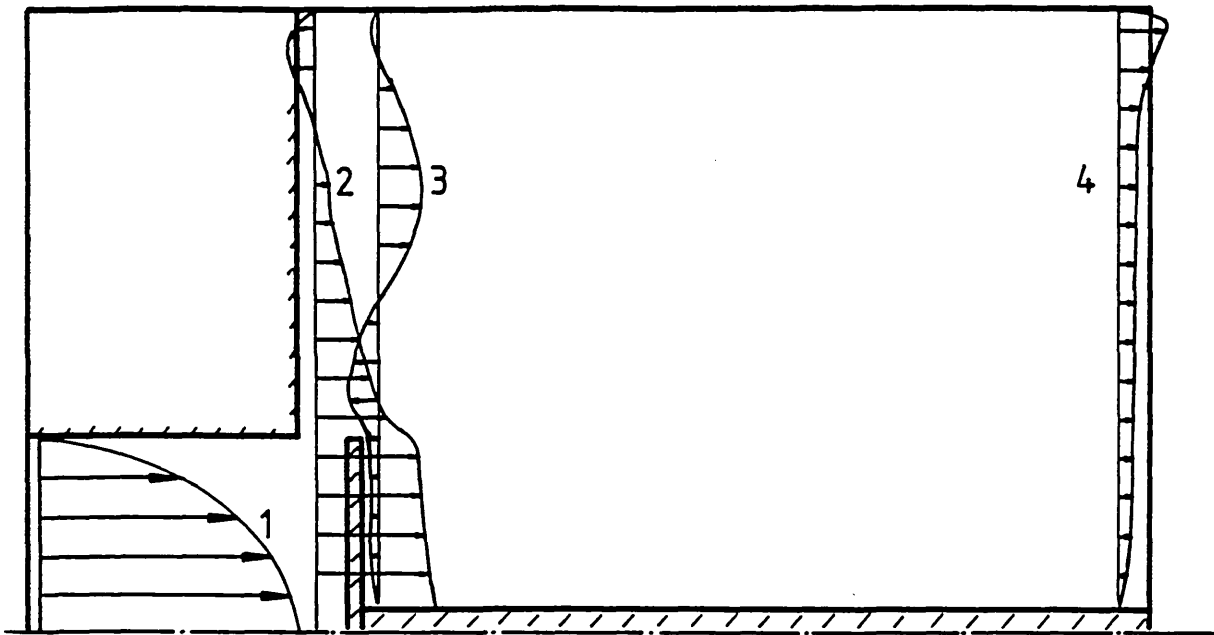


Figure 8.25-3 Axial velocity profiles at four locations:

(1) $Z/D_V = 0.08$; (2) $Z/D_V = 1.188$;

(3) $Z/D_V = L.764$; (4) $Z/D_V = 7.694$;

$Re = 1000$, $L/D = 0.14$, $\theta = 0^\circ$.

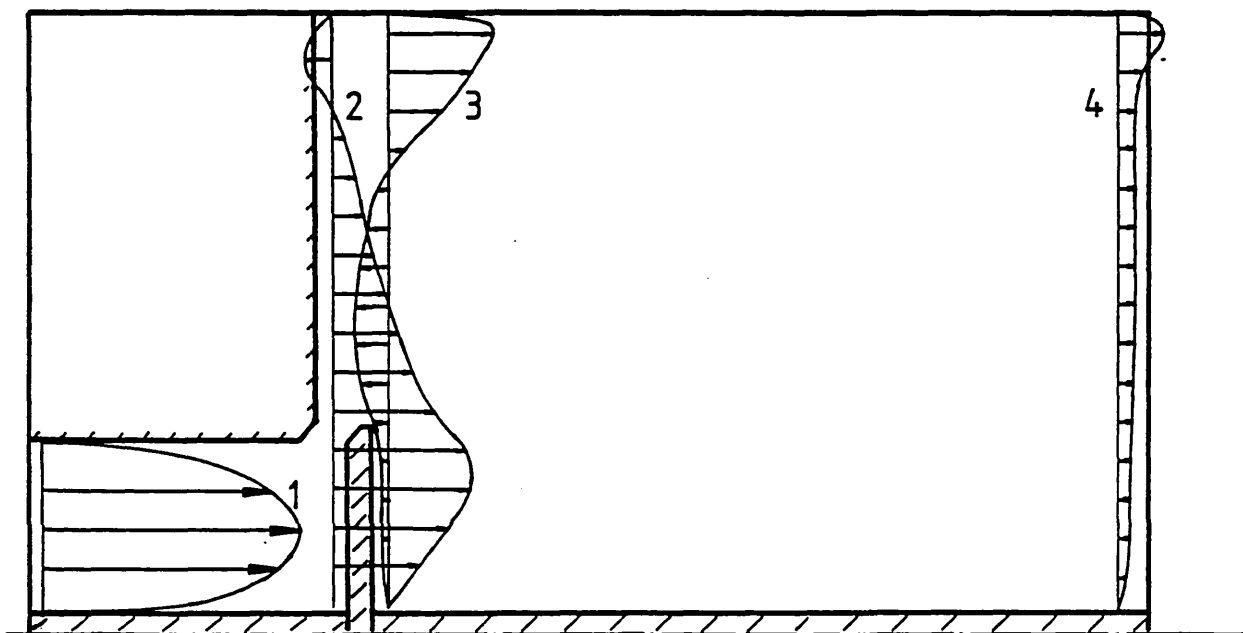


Figure 8.25-4 Axial velocity profiles at four locations:

(1) $Z/D_V = 0.08$; (2) $Z/D_V = 1.188$;

(3) $Z/D_V = 1.764$; (4) $Z/D_V = 7.694$;

$Re = 620$, $L_V/D_V = 0.14$, $\theta = 45^\circ$.

Figure 8.25-5 represents the velocity vector plots for cases 1, 2 and 3 (for a seatless valve).

The streamlines corresponding to three cases shown in Figures 8.25-1 to 8.25-3 are also plotted in Figures 8.25-6 and 8.25-7 with linear interpolation of the stream function, ψ , which is defined as follows:

$$\psi = \int_0^r w r dr \quad , (8.25-1)$$

In Figure 8.25-7 is shown the streamlines for a valve with 45° seat angle, Reynolds number of 620.

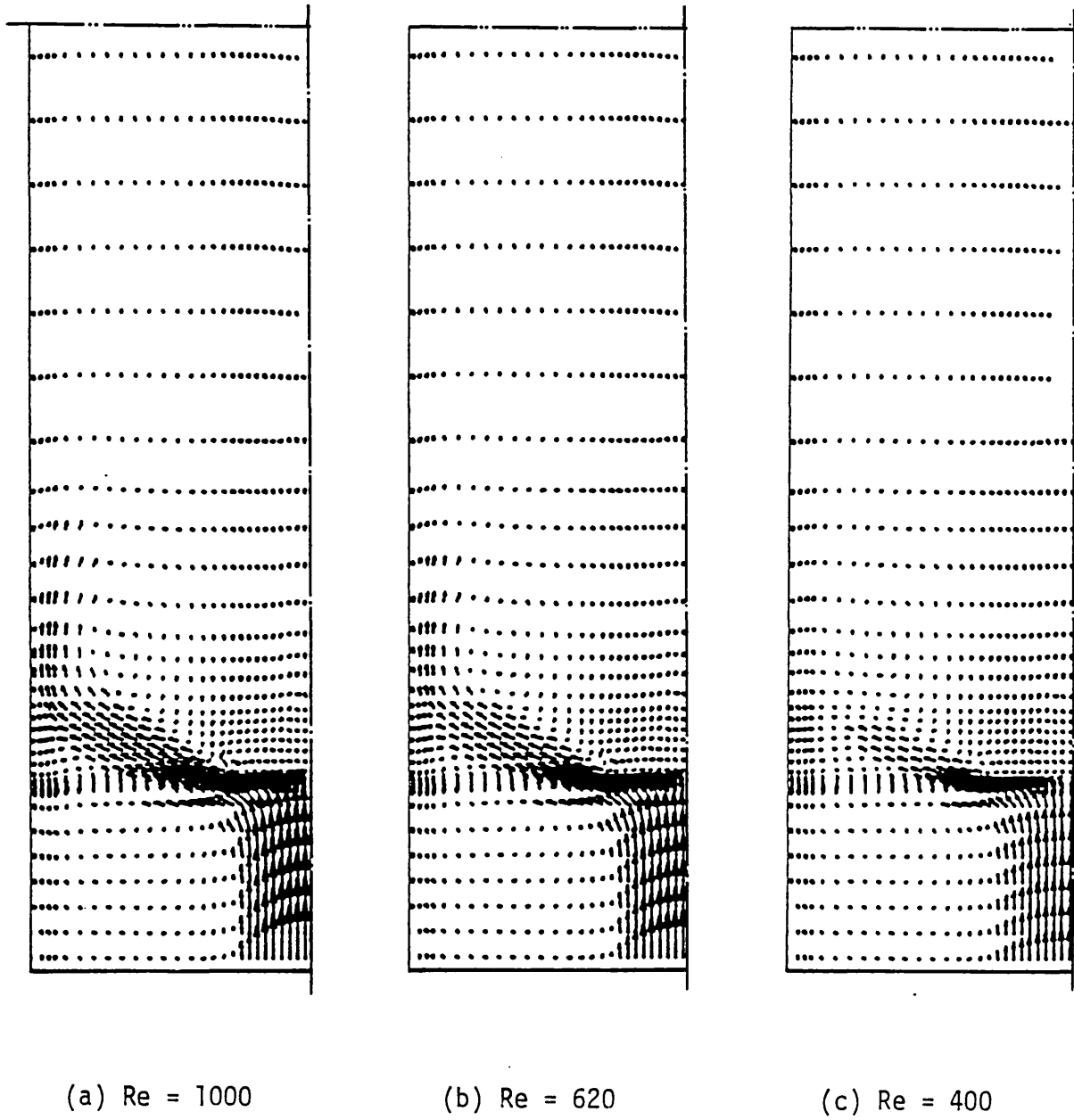


Figure 8.25-5 The predicted velocity vectors for flow through a seatless valve with three different Reynolds numbers of 400, 620 and 1000, $\theta=0^\circ$, $L_v/D_v = 0.14$.

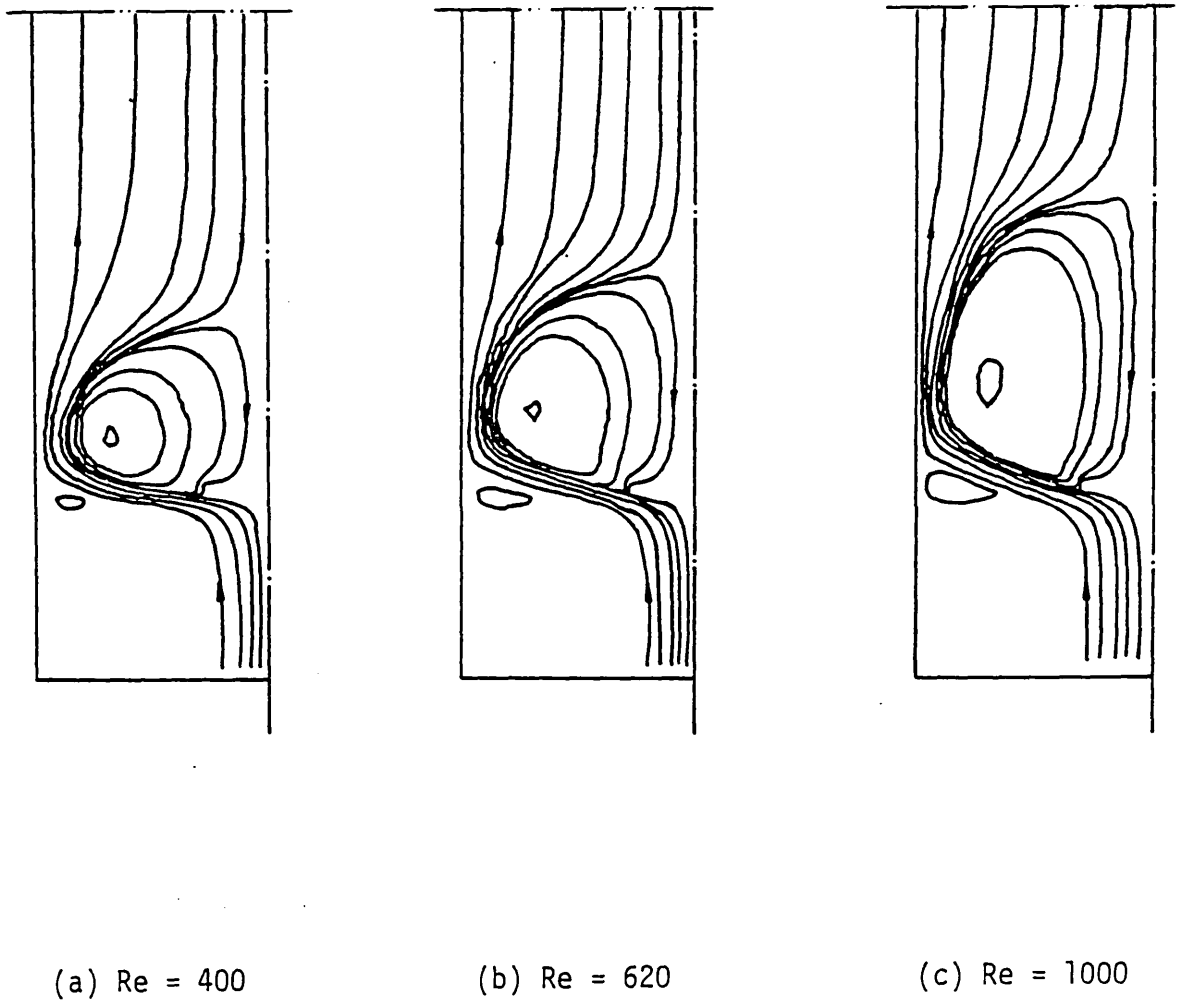


Figure 8.25-3 The predicted streamline pattern near a seatless valve,
for three different Reynolds numbers of 400,620,1000;
($\theta = 0^\circ$, $L_V/D_V = 0.14$).

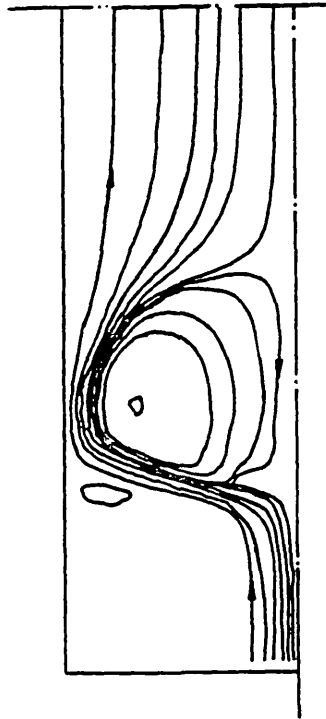


Figure 8.25-7 Predicted streamline pattern near a seated valve.

$Re = 620$, $L_v/D_v = 0.14$, $\theta = 45^\circ$.

It can be seen that the contours form two vortex-rings: a large toroidal vortex just downstream of the valve, and a corner ring trapped between the head and the cylinder wall. When these are compared with the corresponding experimental records of Figure 7.4-1 to 7.4-4, we observe the following. The predicted corner ring is very close to the one on the photographs in shape as well as in position. The main toroidal vortex is also similar to that of the photographs.

The predicted centres of the vortex-rings are compared with those obtained from the experiment in Table 8.25-2 for three different Reynolds numbers.

Table 8.25-2 The predicted and experimentally obtained positions of the vortex cores.

Re	variables	experimental results	present prediction
400	corner vortex Y_c/D	0.413	0.402
	main vortex Y_m/D	0.323	0.332
620	corner vortex Y_c/D	0.417	0.418
	main vortex Y_m/D	0.321	0.332
1000	corner vortex Y_c/D	0.418	0.412
	main vortex Y_m/D	0.321	0.332

In the table, Y_c is the distance from the axis to the centre of the corner vortex. Y_m is that for the main vortex downstream of the valve and D is

the diameter of the cylinder. It is seen that the predicted results for both vortex rings generally agree with the experiments.

8.3 Unsteady-Flow Computations

8.31 The Phenomena to be Predicted

Water from a vertical pipe flows past a disc valve located a short distance downstream of the pipe exit into a coaxial cylinder of a larger diameter. Initially, the cylinder is empty; at subsequent times a water-air interface rises slowly and steadily at a preset rate.

The task here is to predict the velocity distributions and other flow properties at different times.

8.32 Geometry and Dimensions

The main geometry and cylinder dimensions for this flow situation are the same as those for the steady-flow one described in section 8.22. However, the transient nature of the present flow requires that for economy of computer storage the calculation domain should change with time; this point will be elaborated on later in section 8.33.

The flow configuration is as shown in Figure 8.32-1, where the broken-line shows the position of the water-air interface, taken to be the 'initial' position, i.e. at $t = 0$. The water-air interface is taken to be one of the boundaries of the calculation domain; typically, another inlet boundary is chosen to be 0.035 m upstream of the cylinder head (step wall). With the assumption of axial symmetry, the smallest effective

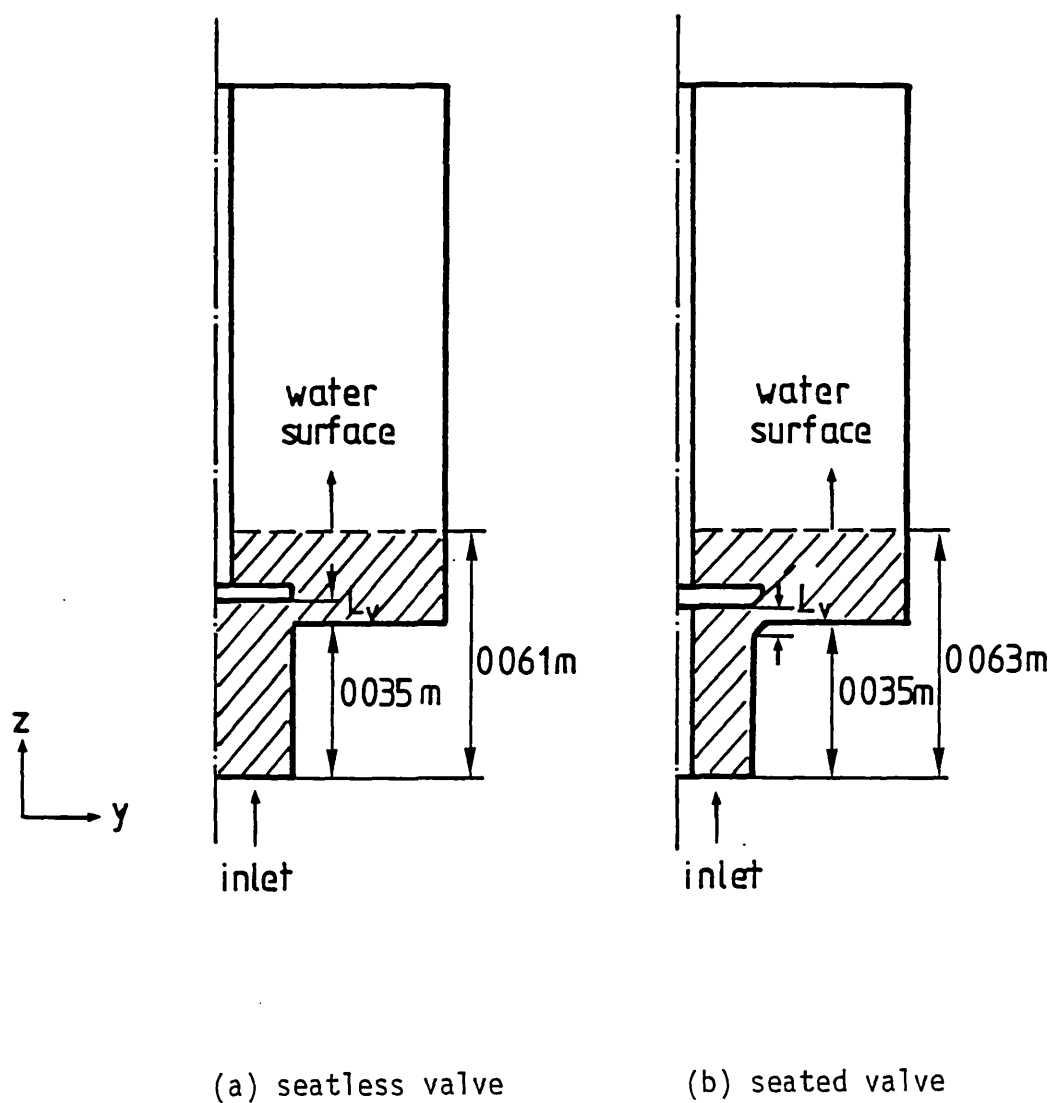


Figure 8.32-1 Unsteady flow through a coaxial valve-port geometry and dimensions.

calculation domain that must be employed is that shown shaded in Figure 8.32-1.

Two selected cases of the experimental situations employed are depicted in Table 8.32-1.

Table 8.32-1 Cases studied and grid employed.

state	Reynolds No Re	valve geometry			grid number NYXNZ
		angle Θ	stem d	lift L/D _v	
unsteady	620	0	0	0.14	27 x 45
	620	45°	0.008	0.14	27 x 48

8.33 The Flow-Domain for Calculation

The calculation domain employed in the present study is shown in Figure 8.33-1. The shaded part is blocked (solid). The water surface, as one boundary, moves with time at a known velocity.

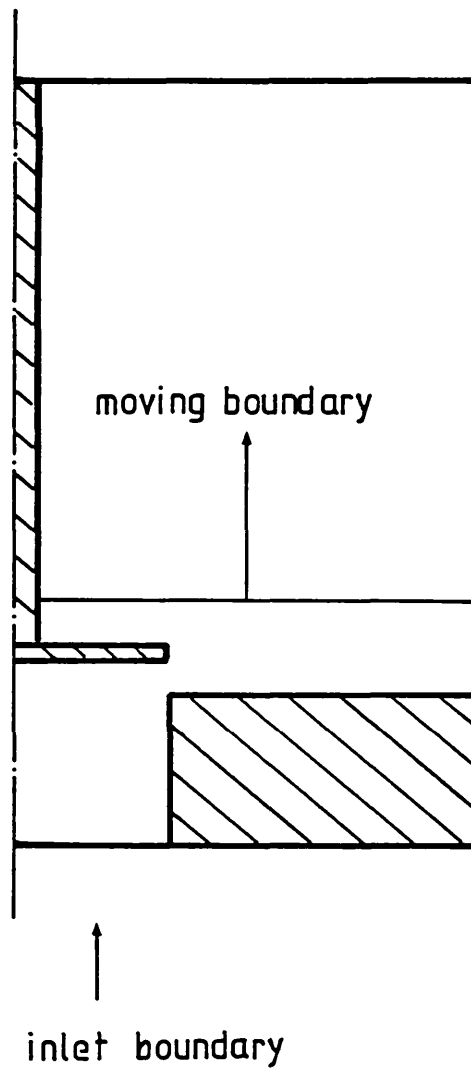


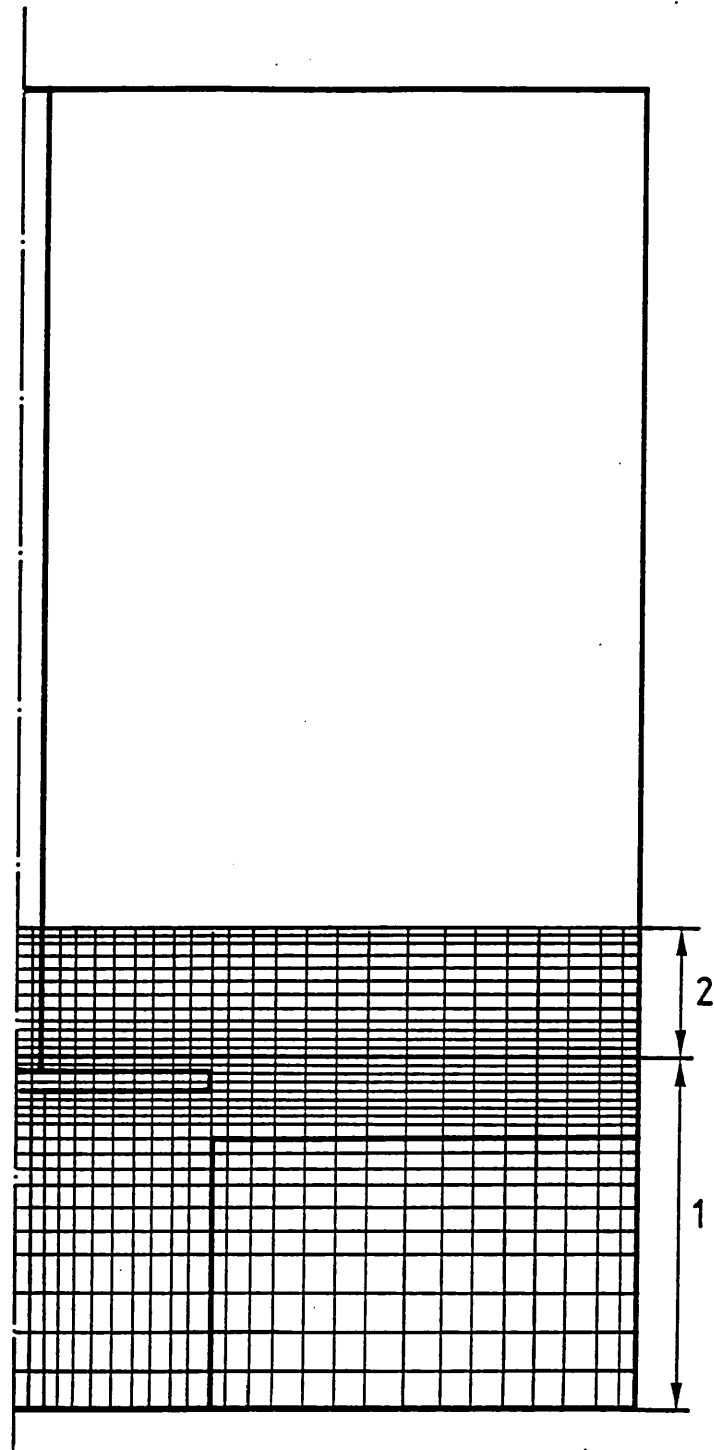
Figure 8.33-1 Flow-domain for calculation.

**8.34 The Use of A Composite Grid, Consisting of A 'Moving' Part and
A Fixed Part**

A feature of the present situation considered is that one of the boundaries of the flow domain, i.e. the water surface, is in motion. Thus, if a fixed computational grid is used, we must have one that covers the highest position of the water surface. Obviously, when the water is at a lower level, this grid is larger than is really needed, giving rise to inefficient use of computer storage and computing time. One way to overcome this difficulty is to allow part of the grid to stretch in sympathy with the movement of the water surface through an appropriate coordinate transformation, while the other part of the grid remains fixed so that the stationary valve assembly can be covered easily.

This method enables the solution domain to be always occupied by the fluid. The accuracy and economy of computational procedure and time are thereby improved. Details of the necessary coordinate transformation is given in Appendix B.

The calculation domain and grid distribution are shown in Figure 8.34-1. The grid is non-uniformly distributed in both axial and lateral directions for all cases.



- (1) fixed grid region
- (2) moving grid region in which the grid in
the axial direction is stretched to follow
the water level

Figure 8-34-1 Two-domains for calculation and grid distribution.

8.35 Boundary and Initial Conditions

The boundaries of the computational domain are as shown in Figure 8.35-1.

Boundary Conditions

Axis of Symmetry

The axis line of symmetry is a zero flux boundary, i.e. at which,

$$V = 0 \text{ and } \frac{\partial w}{\partial y} = 0.$$

Inlet Boundary

The following conditions were imposed at the inlet.

Parabolic distributions of the axial velocity across the inlet duct, which are shown in Figure 8.35-1, were specified for valves without and with a stem. The same formulae (8.24-1 and 8.24-2) were used as for steady flow.

The radial velocity, V , is assumed to be zero.

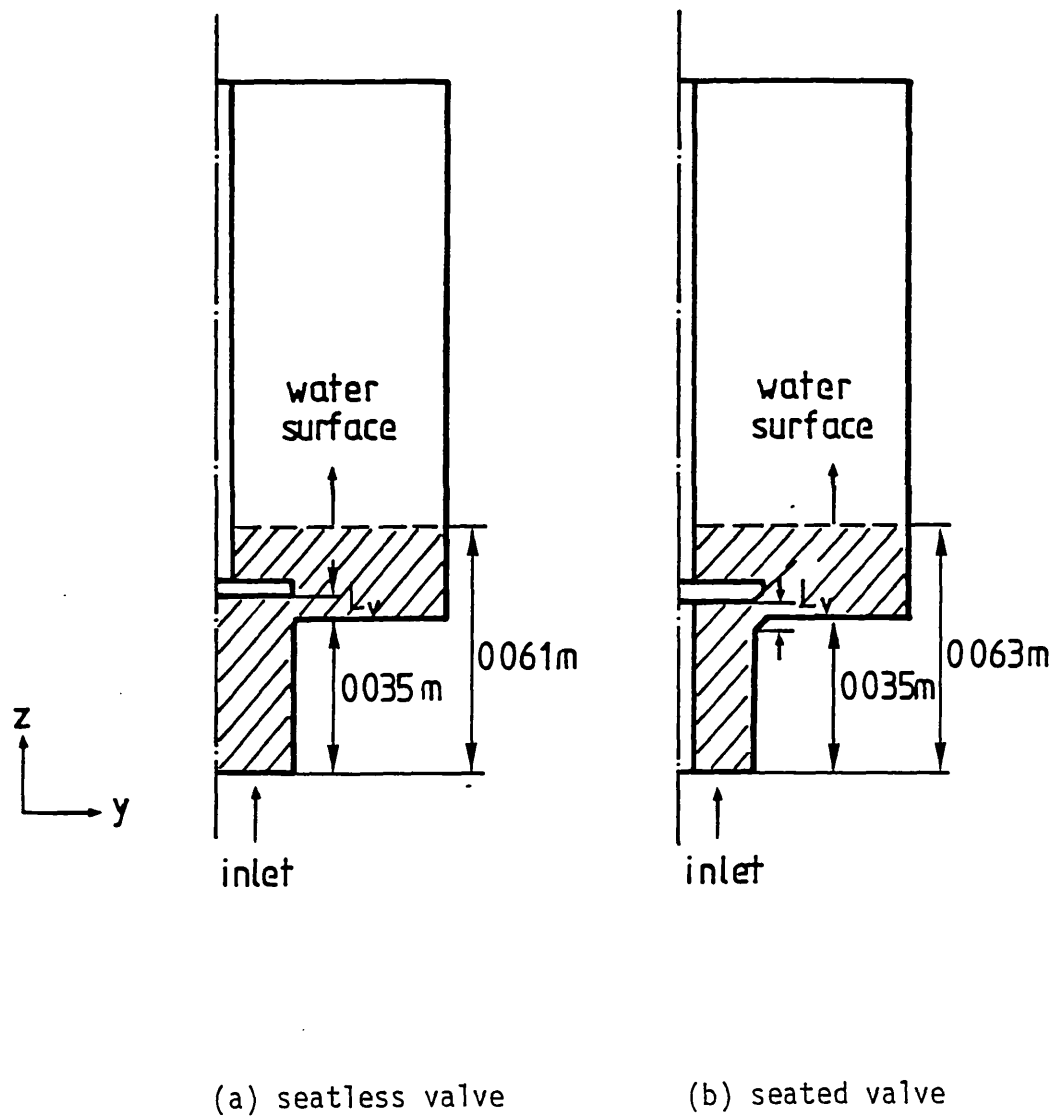


Figure 8.35-1 The boundaries of flow domain.

Free Surface

The free-surface boundary was treated as a moving boundary, which rises with the same velocity as the water surface. The boundary conditions are:

(1) $W = W_S$, where W_S is the rate at which the water surface rises.

$$(2) \frac{\partial V}{\partial z} = 0.$$

Walls

As all walls shown in Figure 8.35-1 were stationary and impervious, both V and W were set to zero on them. Wall functions were employed in all near wall regions.

Initial conditions

The calculation was started from $t = 0$ when the water level was at a height of 0.063m; the initial field values for W , V , and P , at that moment were taken to be those obtained from the steady flow solutions of the initial domain. For this steady-flow calculation all the boundary conditions except the free surface boundary were the same as described above; at the free surface, a constant-pressure condition was imposed.

8.36 Results and Discussion

The flow development during the rising of the water surface is described in several series of diagrams. In Figures 8.36-1 (1) to (4) are shown

the predicted distributions of the axial-velocity of the flow at four locations for a seatless valve with a valve support on the downstream side at four time intervals; those for a seated valve with a valve stem and support on each side of the valve are shown in Figures 8.36-2 (1) to (4). The velocity values have been normalized by the mean velocity at Inlet (w_{In}) of 0.0124m/s or 0.0155m/s respectively for seated and seatless valves.

From Figure 8.36-1 a parabolic profile is seen near the entrance, at $Z/D_v = 0.08$, i.e. profile 1. Profile 2, just downstream of the step but upstream of the valve at $Z/D_v = 1.188$, shows that the velocity changes sign near the cylinder wall, indicating recirculation there, caused by the sudden enlargement. Profile 3, at a section downstream of the valve, at $Z/D_v = 1.764$, indicates the presence of the second recirculation, just downstream of the valve. Profile 4, its position changes with time, i.e. further downstream of the valve and near the water surface, reveals that the axial velocity is larger near the cylinder wall, which is influenced by the upstream flow, but the values at the remaining part of that plane is almost uniform.

Profiles 1 to 3 at four different time intervals shown in Figures 8.36-1 (1) to (4) are similar; profiles 4 in Figures 8.36-1 (1) to (3) are also similar. However, profile 4, in (4) is different from previous ones, the negative values appear near the cylinder wall, which indicates a third recirculation zone there.

From these velocity profiles it is clear that the flow in the cylinder consists of a system of three vortices: a ring vortex at the corner between the step and the cylinder wall, a main vortex occupying most of the flow space downstream of the valve and a third one just beneath the

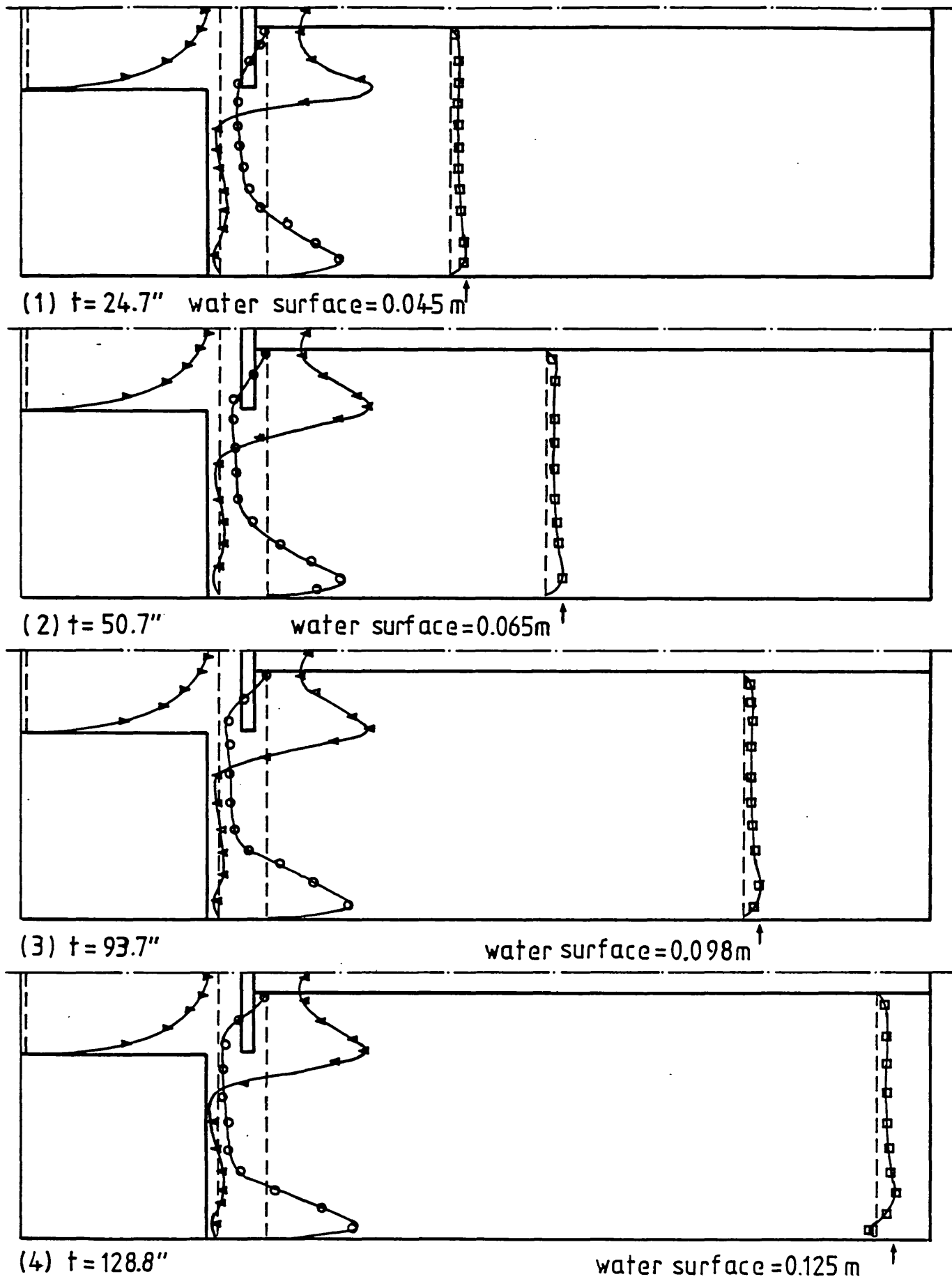


Figure 8.36-1 Axial-velocity profiles at different-time intervals for a seatless valve; $Re = 620$, $L_v/D_v = 0.14$, $\theta = 0^\circ$,
 $\blacktriangleright$ at $Z/D_v = 0.08$, $\blacktriangleleft$ at $Z/D_v = 1.188$,
 $\circ$ at $Z/D_v = 1.764$, $\square$ near water surface.

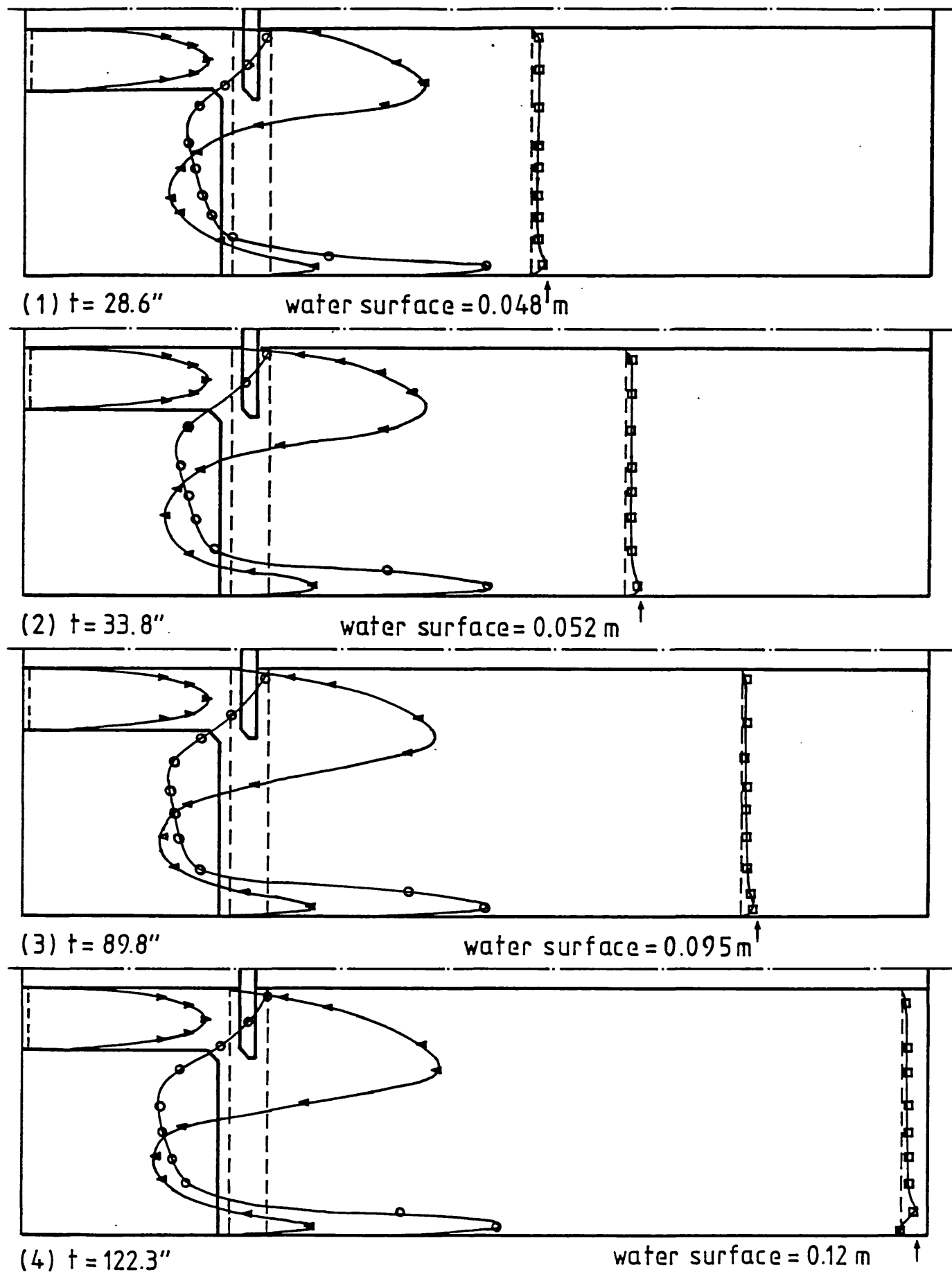


Figure 8.36-2 Axial-velocity profiles at different-time intervals for a valve with 45 deg angle, $Re = 620$, $L_v/D_v = 0.14$, $\theta = 45^\circ$,
 $\blacktriangleright$ at $Z/D_v = 0.08$, $\blacktriangleleft$ at $Z/D_v = 1.188$,
 $\circ$ at $Z/D_v = 1.764$, $\square$ near water surface.

water surface. They are qualitatively in accord with the photographic records of the corresponding experiments.

In Figure 8.36-2 are shown the velocity profiles for a seated valve with a stem on the upstream side of the valve. Because of the extra stem, the velocity profile at the inlet has a different shape, beyond the step wall all the profiles have the same characteristics as those discussed above for the seatless valve.

Figures 8.36-3 and 8.36-4 display the development of the main vortex at the same four time intervals as those in Figures 8.36-1 to 8.36-2 using a particle-tracking technique (see Appendix C for details).

Table 8.36-1 shows the predicted and measured position of the vortex centre. The measured positions have been obtained from photographic records of the corresponding flows, Figures 7.5-8 and 7.5-9.

In the present study particles originated from a given location at a short distance (0.009 m) below the cylinder bottom, and moved at the end of each time step according to the velocity computed for the flow domain; each particle can then be identified by the time it passes through the injection point, thus the filament line of particles represents the trace of all the particles originating from that point during the time between $t = 0$ and certain time interval $t = t_i$.

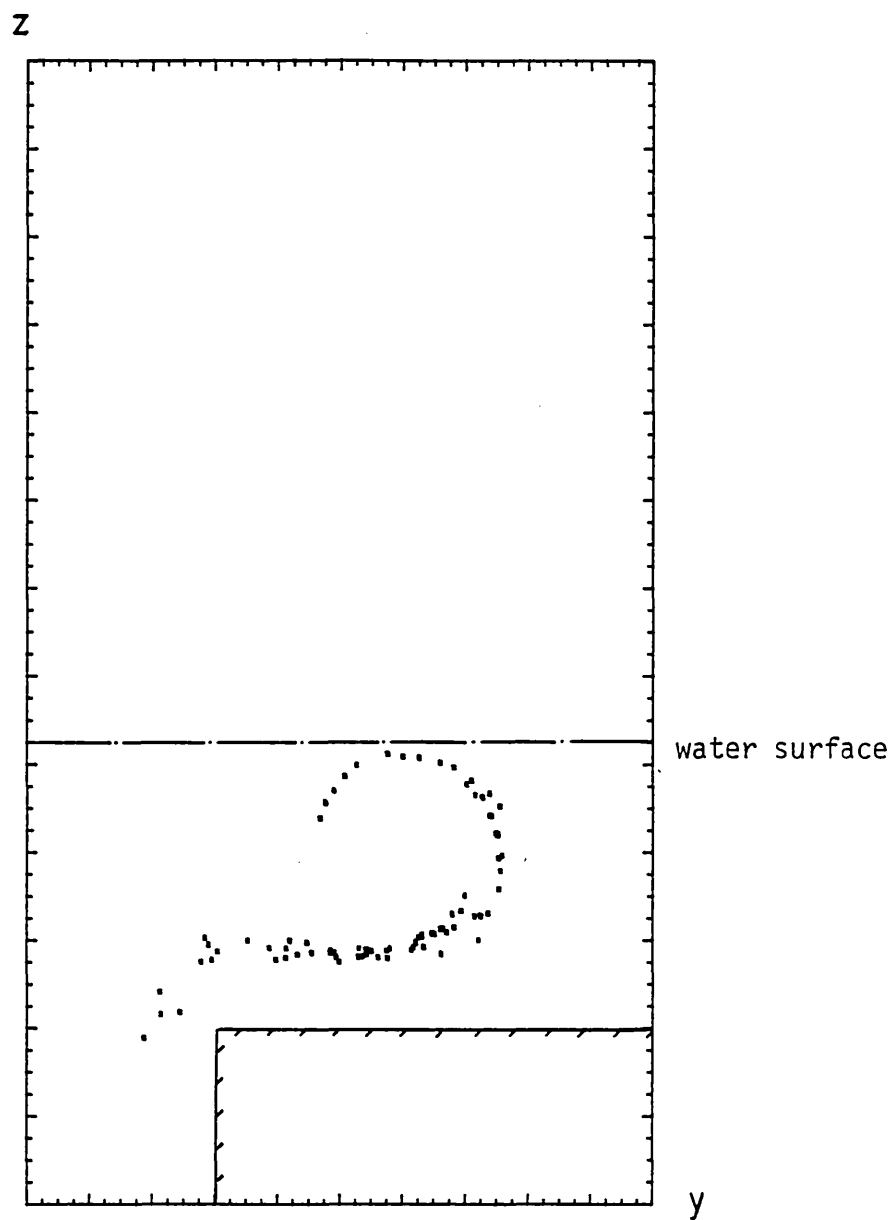


Figure 8.36-3 Filament of dye particles for a seatless valve.

$Re = 620$, $L_v/D_v = 0.14$, $\Theta = 0^\circ$.

(1) between $t = 0$ and $t = 24''$

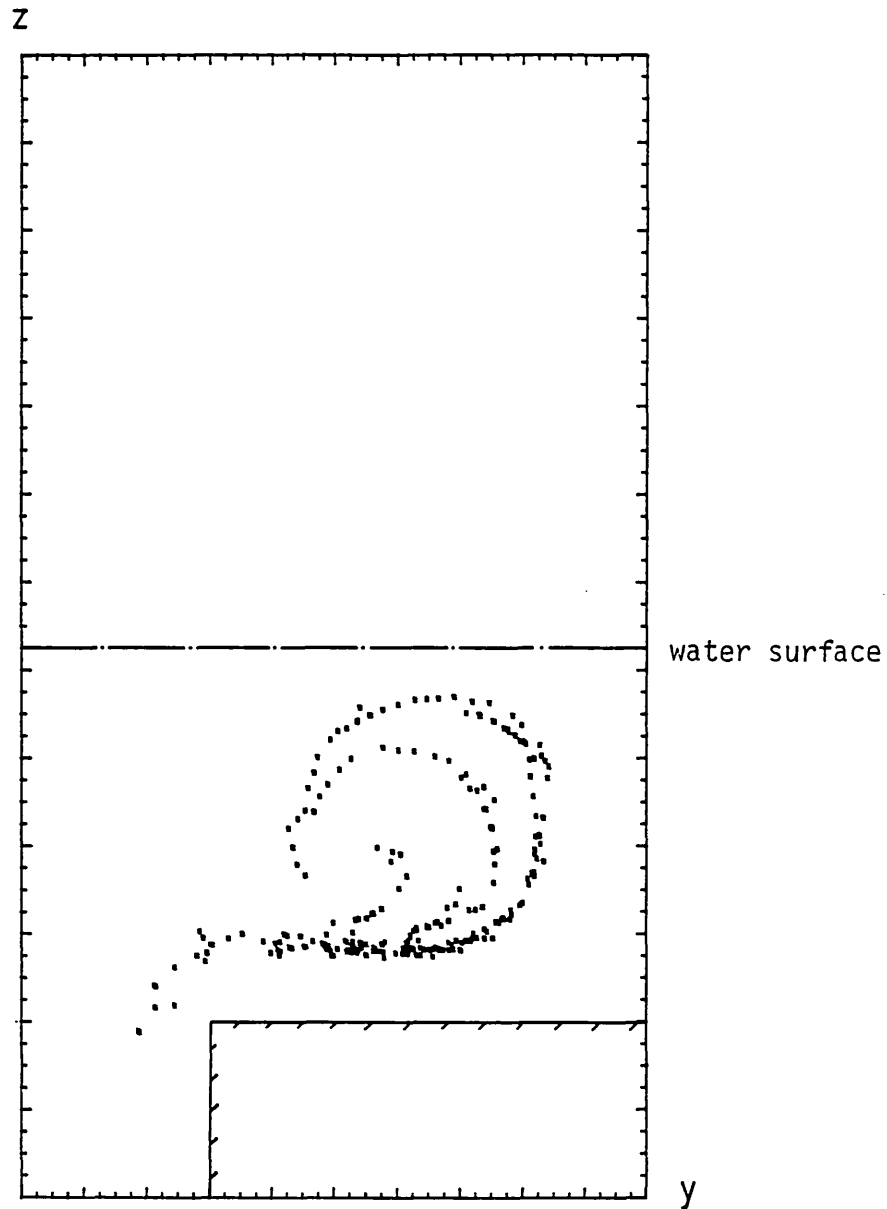


Figure 8.36-3 Filament of dye particles for a seatless valve.

$Re = 620$, $L_v/D_v = 0.14$, $\Theta = 0^\circ$

(2) between $t = 0$ and $t = 50.7^\circ$

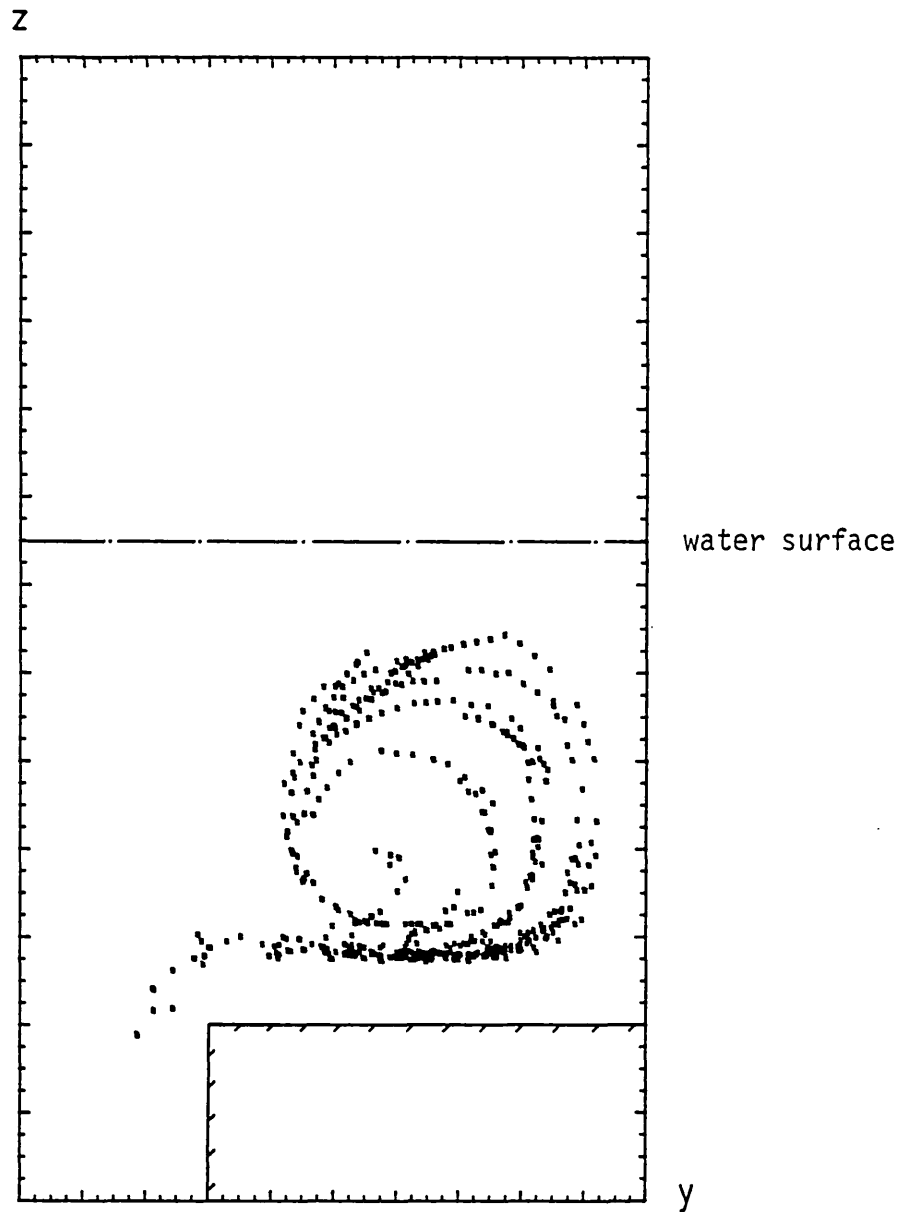


Figure 8.36-3 Filament of dye particles for a seatless valve.

$Re = 620$, $L_v/D_v = 0.14$, $\Theta = 0^\circ$

(3) between $t = 0$ and $t = 93.7^\circ$.

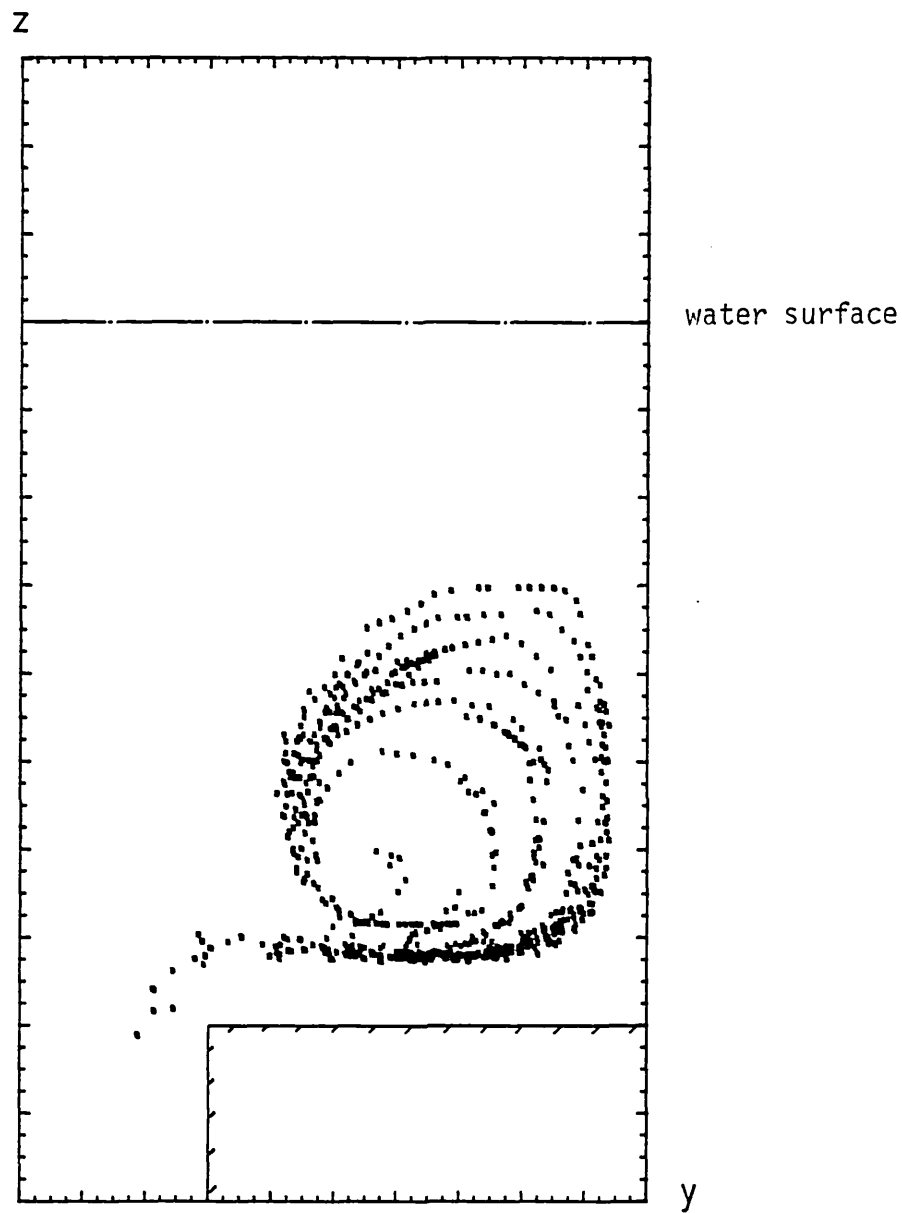


Figure 8.36-3 Filament of dye particles for a seatless valve.

$Re = 620$. $L_v/D_v = 0.14$. $\theta = 0^\circ$.

(4) between $t = 0$ and $t = 128.8^\circ$.

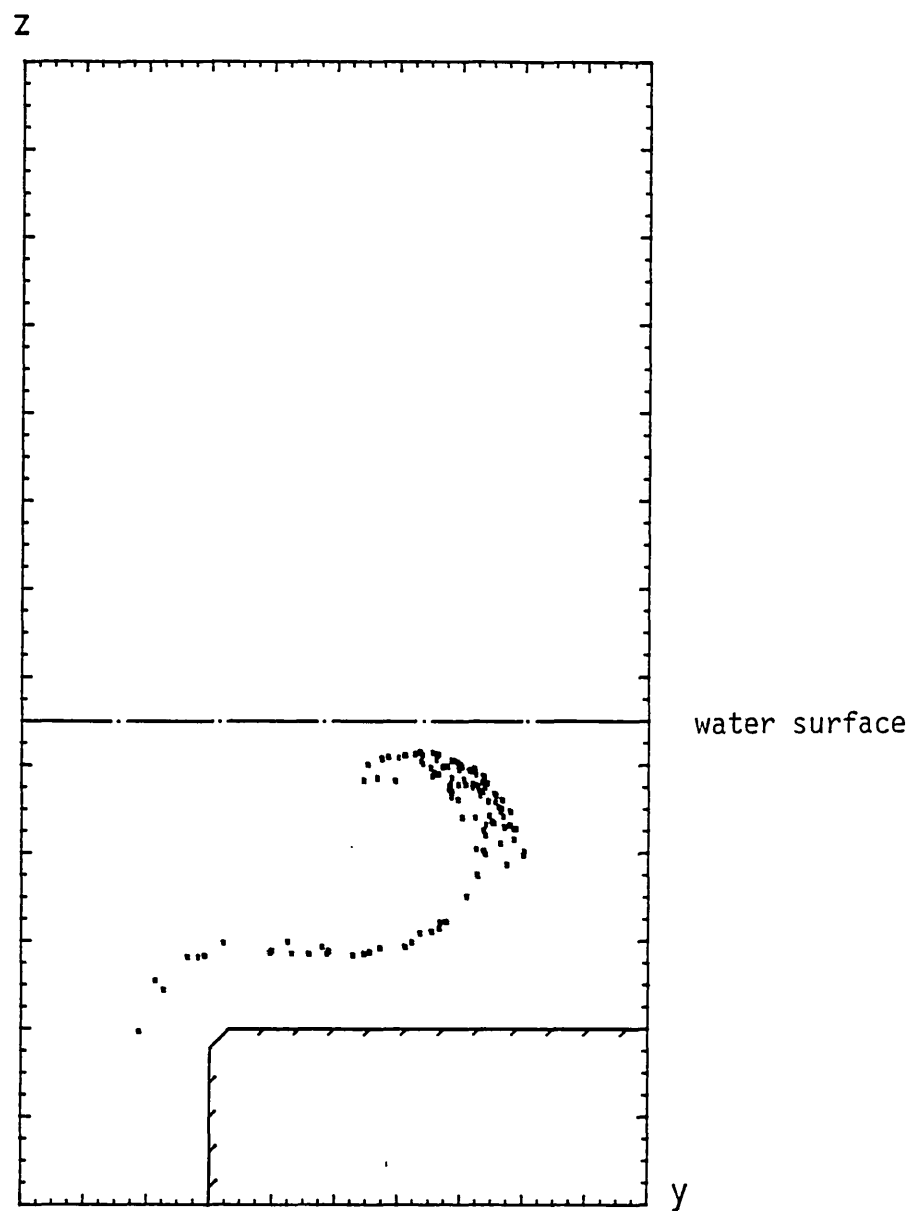


Figure 8.36-4 Filament of dye particles for a seated valve.

$Re = 620$ $L_v/D_v = 0.14$. $\Theta = 45^\circ$.

(1) between $t = 0$ and $t = 40^\circ$.

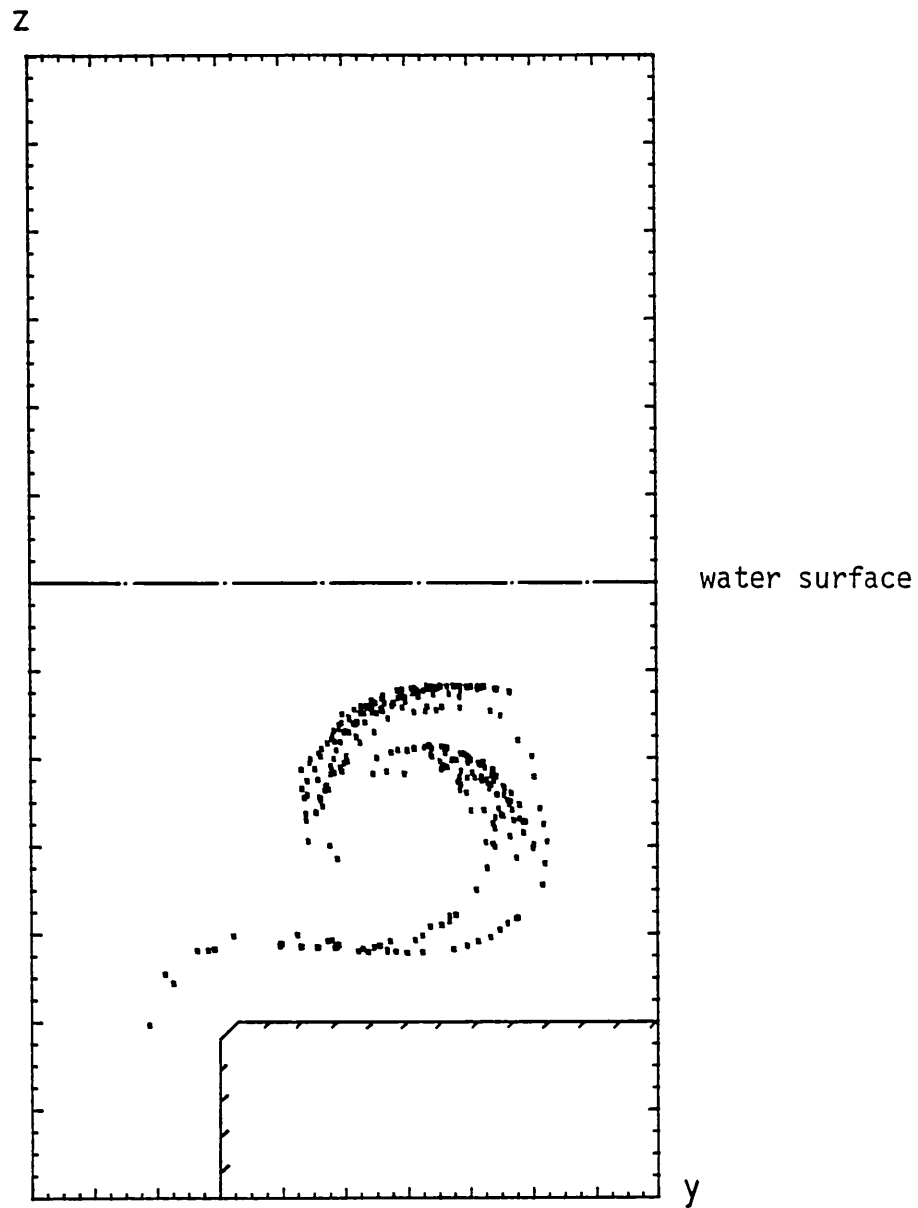


Figure 8.36-4 Filament of dye particles for a seated valve.

$Re = 620$ $L_v/D_v = 0.14$. $\Theta = 45^\circ$.

(2) between $t = 0$ and $t = 48^\circ$.

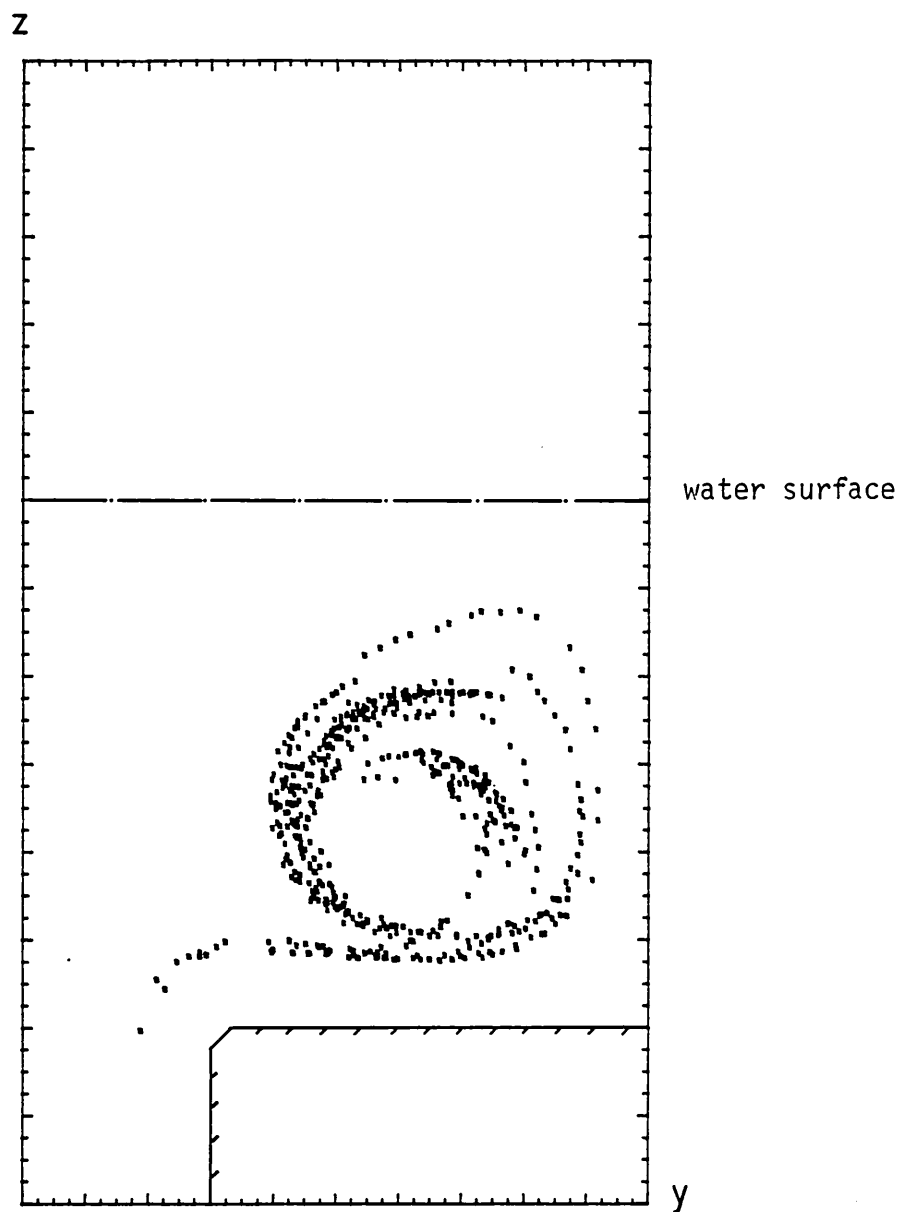


Figure 8.36-4 Filament of dye particles for a seated valve.

$Re = 620$ $L_v/D_v = 0.14$, $\Theta = 45^\circ$.

(3) between $t = 0$ and $t = 64^\circ$.

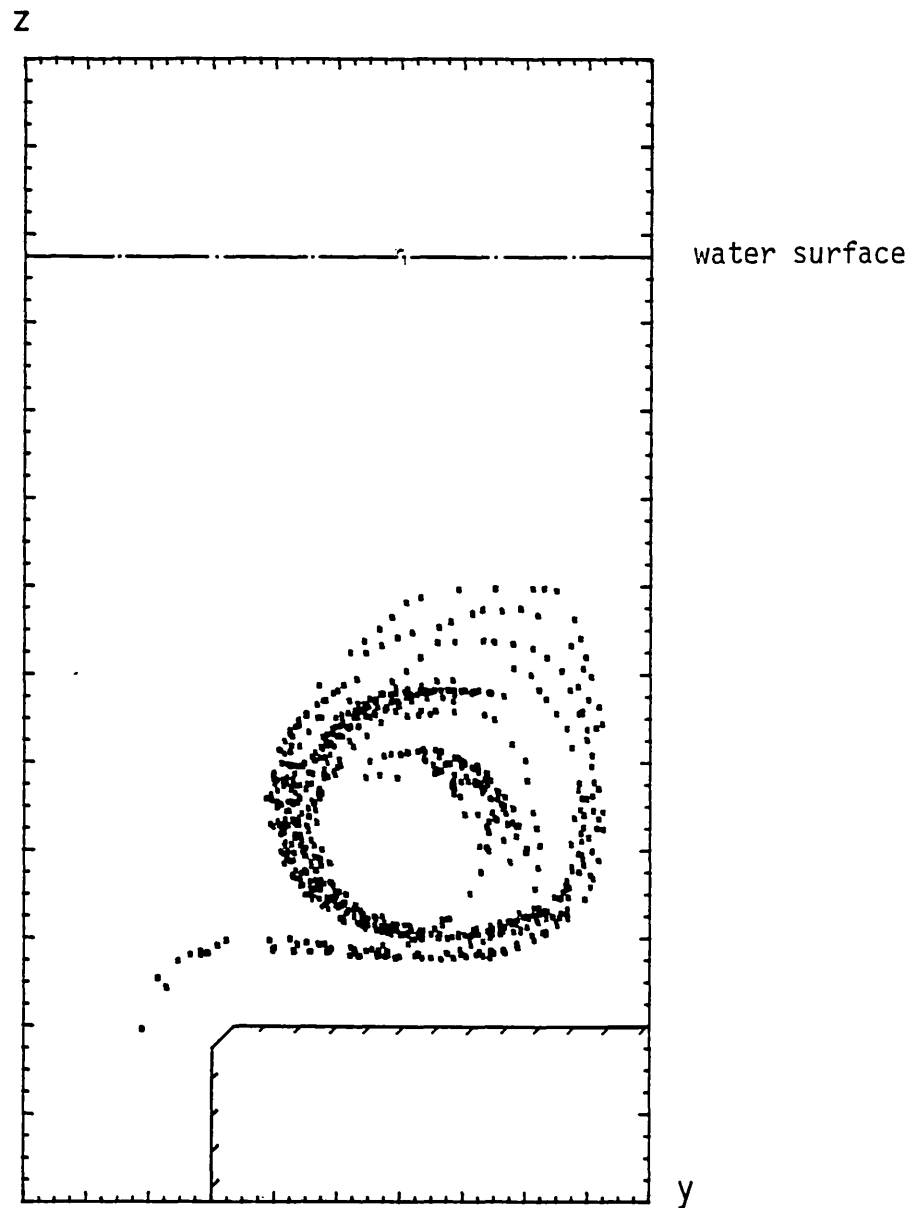


Figure 8.36-4 Filament of dye particles for a seated valve.

$Re = 620$, $L_v/D_v = 0.14$, $\theta = 45^\circ$

(4) between $t = 0$ and $t = 112^\circ$.

It is seen in Figures 8.36-3 and 8.36-4 that only the main vortex is shown because no particle appears either at the corner or near water-air interface region. The development of the main vortex downstream of a seatless valve, as shown in Figure 8.36-3(1) to (4) is discussed here in detail. (1) represents the filament line of particles passing through the injecting point during the time between $t = 0$ and $t = 27.7''$ seconds, (2) for time between $t = 0$ and $t = 50.7''$, (3) for time between $t = 0$ and $t = 93.7''$, and (4) for time from 0 to $128.8''$. They are compared with the corresponding experimental records of Figures 7.5-8 and 7.5-9 which are the instantaneous loci of all dye particles passed injection points for the corresponding time intervals. The general shape and the initial trajectory angle of the main vortex downstream of the valve are close to the measurement from the experiments. (Figure 7.5-8) (1) to (4)).

It can also be seen that the agreement between the predicted and measured positions of the vortex center is generally satisfactory, the maximum discrepancy between predictions and measurements being less than 9%. The agreement between predictions and measurement from the experiment could be considered generally acceptable.

Table 8.36-1 Measured and predicted positions of the vortex centres

		experimental value (y/D, Z/D)	present prediction (y/D, Z/D)
I.	corner vortex	(0.393, 0.161)	(0.361, 0.147)
	main vortex (downstream of the valve)	(0.208, 0.184)	(0.189, 0.168)
II.	corner vortex	(0.416, 0.242)	(0.393, 0.235)
	main vortex	(0.242, 0.248)	(0.219, 0.228)
III.	corner vortex	(0.393, 0.126)	(0.367, 0.118)
	main vortex	(0.306, 0.266)	(0.283, 0.251)
IV.	main vortex	(0.295, 0.353)	(0.282, 0.334)

8.4 Closure

It has been demonstrated in this chapter that numerical procedure can be used for simulating the experimental flow. The predicted vortices have been compared with the experimental observations and agreement has been found to be fair. Predicted profiles of the axial velocity have also been presented for both steady and unsteady flow; they all appear to be realistic.

CHAPTER 9 - CONCLUSIONS

This, the final chapter of the thesis, provides a summary of the main conclusions of the present theoretical and experimental investigation in section 9.1 and 9.2. Section 9.3 contains some suggestions for future research.

9.1 Theoretical Predictions

Suitable SATELLITE-GROUND attachments to the PHOENICS computer code have been set up to simulate several complex flow problems. Particular attention has been paid to the flow around an inlet valve and systematic studies of this flow have been made.

Because well-documented data are available in the literature, the flow through an axisymmetric expansion has been used to validate the turbulence model and wall functions embodied in the PHOENICS code and to check the accuracy of using porosity as a means of handling boundaries which are not parallel to the grid lines. As a result it is concluded that the numerical procedure and turbulence model employed are realistic, and the porosity method is adequate for the simulation of inclined walls.

Turbulent air flows through several poppet valves, both seatless and seated ones, have been simulated, and details of the flow field examined. The predicted distributions of flow properties like velocity, pressure and turbulence energy all appear to be realistic. Unfortunately, no suitable experimental data are available for comparison. However, a

quantitative comparison can be made of the predicted values of the discharge coefficient with the measurements of Annands's experiments. The fact that fair agreement has been obtained for this for the whole range of the parameters considered, can be taken as an indication that the computational model as a whole is adequate for predicting the complex flows around valves.

The particle-tracking technique used has been found to be very useful in tracing the fluid filament-lines in an unsteady flow allowing an easy interpretation of the flow patterns.

The computing resources required for all the computations made have been moderate. It is however, expected that very much more computing resources will be used for three-dimensional flows with the effects of piston and valve movement fully accounted for.

9.2 Experiments

A water-model rig has been constructed and used to study the fundamentals of the valve flow. A flow-visualization technique has been used to reveal some important details of the laminar flow around the valve. The flow structures for both steady and unsteady flows through the valve have been observed and recorded. Comparison between the observed flow patterns around the valve with the corresponding predictions show reasonable agreement.

A circumferential motion was repeatedly observed throughout the tests, both steady and unsteady ones. This "pseudo-diffusion" in the circumferential direction within the cores of the toroidal vortices cannot

be accounted for reliably with the limited information available, but an explanatory theory has been essayed.

9.3 Suggestion for Future Work

In order to provide a more complete understanding of the fluid flow through poppet valves, a few areas for future research are suggested as follows:

Experiments

- (1) More measurements of the flow properties (e.g. velocity, pressure, turbulence quantities) should be made under steady conditions.
- (2) Further measurements of the flow properties under controlled conditions resembling real-engine situations should be made.
- (3) More quantitative measurements of the flow field should be made in the water-model rig to allow more insight into the origin of the circumferential motion.

Computations

- (1) Further computations of the present kind should be made with simple movement of the valve and piston included.
- (2) Some means of expressing the circumferential motion phenomenon in mathematical terms should be devised in order that computer simulation of such processes should be realistic.
- (3) The use of "porosity" in simulating complex geometries is not particularly economical in computing time resources. More

economical alternatives, like body-fitted coordinates, should be tested.

REFERENCES

1. AHMADI-BEFRUI B., GOSMAN A.D., LOCKWOOD F.C. & WATKINS A.P. (1981)
'Multidimensional Calculation of Combustion in an Idealised Homogeneous Charge Engine: a Progress Report'.
SAE Paper 810151.
2. AHMADI-BEFRUI B., ARCOUMANIS C., BICEN A.F., GOSMAN A.D., JAHANBAKHSI A. & WHITEHEAD J.H. (1982).
'Calculation and Measurements of the Flow in a Motored Model Engine and Implications to Open-Chamber Direct-Injection Engines'.
Proc. Symp. on Three Dimensional Turbulence Shear Flows, St. Louis.
3. ANNAND W.J.D. (1969)
'Engine Breathing'.
Automobile Engineer, 59.
4. ANNAND W.J.D. & ROE G.E. (1974)
'Gas Flow in the Internal Combustion Engine'.
Academic Press.
5. ARCOUMANIS C. & VLACHOS N.S. (1980)
'The Influence of Valve Seat Angle on the Flow Patterns in a Motored Piston-Cylinder Assembly'.
Imperial College, Mech. Eng. Dept. Report FS/80/25.

6. ARCOUMANIS C., BICEN A.F., VALCHOS N.S. & WHITELAW J.H. (1982A)
'Effects of Flow and Geometry Boundary Conditions on fluid Motion in a Motored I.C. Model Engine',
Proc. I. Mech. E. 196, 4.
7. ARCOUMANIS C., BICEN A.F. & WHITELAW J.H. (1982C)
'Effects of Inlet Parameters on the Flow Characteristics in a Four-Stroke Model Engine',
SAE Paper 820750.
8. ARNOLD M.J., TINDAL M.J. & WILLIAMS T.J. (1972)
'Measurements of Induction Gas Velocities in a Reciprocating Engine Cylinder',
SAE paper 720115.
9. ASANUMA T. & OBOKATA T. (1979)
'Gas Velocity Measurements of a Motored and Firing Engine by Laser Anemometry',
SAE paper 790096.
10. ASMUS T.W. (1982)
'Valve Events and Engine Operation',
SAE Paper 820749.
11. AWA A.G.A. (1979)
'Numerical Modelling of Flows with Moving Interfaces',
Ph.D Thesis. University of London.

12. AWA A. G. , SPALDING D. B. & SRIVATSA S. K. (1977)
'Air Flow in Diesel-Engine Cylinders',
HTS/77/30.
13. BEER J. M. & DAVIES T. W. (1971)
'Flow in the Wake of Bluff-Body Flame Stabilisers',
In Proc. 13th Symp. (Intl) on Combustion, pp 631-638.
Combustion Institute, Pittsburgh.
14. BENSON R. S. (1960)
'Experiments on a piston controlled port',
The Engineer 210, 1960, p. 875.
15. BENSON R. S. , GARG R. D. & WOOLLATT D. (1964)
'A Numerical Solution of Unsteady Flow Problems',
Int. J. Mech. Sci. 6, p. 117, 1964.
16. BERTODO R. & SIRAKUMARAN S. (1973)
'An Assessment of Diesel Engine Poppet Valves',
Proc. I. Mech. E. 187, p31, 1973.
17. BERTODO R. & SIRAKUMARAN S. (1973)
'An Assessment of Diesel Engine Poppet Valves',
Proc. I. Mech. E. 187, p. 31, 1973.
18. BETTS P. L. & YUE Y. K. (1970)
'Vortex Flow in a Cylinder',
I. Mech. E. Thermo & Fluids Convention, Glasgow, p. 201,
1970.

19. BICEN A. F. (1983)
'Air Flow Characteristics of Model Internal-Combustion Engines',
Ph.D Thesis, University of London.
20. BLAIR G. P. & ARBUCKLE J. A. (1970)
'Unsteady Flow in the Induction System of a Reciprocating Internal
Combustion Engine',
SAE paper 700493.
21. BORMAN G. L., MYERS P. & UYEHORA O. (1972) A1
'Engine Simulation Studies',
OSTACOM Report # 11398, University of Wisconsin, March 1972.
22. BRADSHAW P. (1973)
'Effects of Streamline Curvature on Turbulence Flow',
AGARDograph 169.
23. BRANDI F., REVERENCIC I. CARTELLIERI W., & DENT J. C.
(1979)
'Turbulent Air Flow in the Combustion Bowl of D.I. Diesel Engine
and its Effect on Engine Performance',
SAE paper 790040.
24. CHAO S. P., FUNK J. E. & WOOD D. J. (1972)
'The Transient Response of Orifices and Very Short Lines'
ASME paper No. 71-WA/FE-14.

25. CHATURVEDI M.C. (1963)
'Flow Characteristics of Axisymmetric Expansion',
ASCE, J. Hydraulics Division, 89, p.61.
26. CHIGHER N.A. & GILBERT J.L. (1968)
'Recirculation Eddies in the Wake of Flameholders',
J. Inst. Fuel, 41, 105-113.
27. COLE B.N. & MILLS B. (1953)
'The Theory of Sudden Enlargements Applied to the Poppet Exhaust Valve',
Proc. I. Mech. E. 1B p.364, 1953.
28. DANESHYAR H., FULLER D.E. & DECKKER B.E.L. (1973)
'Vortex Motion Induced by the Piston of an Internal Combustion Engine',
Int. J. Mech. Sci. 1973. Vol 15, pp 381-390.
29. DENT J.C. & SALAMA N.S. (1975)
'The Measurement of the Turbulence Characteristics in an Internal Combustion Engine cylinder',
SAE paper 750886.
30. DENNISON E.S., KUCHLER T.C., & SMITH D.W. (1931)
'Experiments on the Flow of Air Through Engine Valves',
Trans. Amer. Soc. Mech. Engrs (Oil Gas Pwr) 1931 53
(OGP-53-6), 79.

31. DENT J.C. & DERHAM J.A. (1974)
'Air Motion in a Four-Stroke Direction-Injection Diesel Engine',
Proc. I. Mech. E., 188, 269.
32. EKCHIAN A. & HOULT D.P. (1979)
'Flow Visualization Study of the Intake Process of an Internal
Combustion Engine',
SAE Paper 790095.
33. EL-TAHRY S.H. (1982)
'A Numerical Study on the effects of Fluid Motion at Inlet-Valve
Closure on Subsequent Fluid Motion in a Motored Engine',
SAE Paper 820035.
34. FUKUTANI I. & WATANABE (1982)
'Air Flow Through Poppet Inlet Valves - Analysis of Static
and Dynamic Flow Coefficients',
SAE Paper 820154.
35. GOLDSTEIN S. (ed., 1938)
'Modern Development in Fluid Dynamics',
Vol. 1,2, Dover.
36. GOSMAN A.D., PUN W.M., RUNCHAL A.K., SPALDING D.B. &
WOLFSHTEIN M. (1969)
'Heat and Mass Transfer in Recirculating Flows',
Academic Press.

37. GOSMAN A.D. & WATKINS A.P. (1976)
'Simulation of Flow and Heat Transfer During a Four-Stroke Motored Engine Cycle',
Imperial College, Mech. Eng. Dept. Report.
38. GOSMAN A.D., MELLING A., WATKINS A.P. & WHITELAW J.H. (1977)
'Axisymmetric Laminar Flow in a Motored Reciprocating Engine',
Imperial College, Mech. Eng. Dept., Report CHT/77/3.
39. GOSMAN A.D. & JOHNS R.J.R. (1978)
'Development of a Predictive Tool for In-Cylinder Gas Motion in Engines',
SAE Paper 780315.
40. GOSMAN A.D., MELLING A., WHITELAW J.H. & WATKINS A.P. (1978b)
'Axisymmetric Flow in a Motored Reciprocating Engine',
Proc. I. Mech. E., 192, 11.
41. GOYAL M., SCHARPF GLEN & BORMAN G. (1967)
'The Simulation of Single Cylinder Intake and Exhaust Systems',
SAE Paper 670478.
42. GRASSO F. & BRACCO F.V. (1981)
'Computed and Measured Turbulence in Axisymmetric I.C. Engines',
Princeton University, Mech. and Aerospace Eng. Dept. Report,
also submitted to AIAA Journal.

43. HARTLEY J. (1976).
'Engine Design Series-Valve Gear',
IMechE 1976 Automotive Engineer.
44. HINZE J. O. (1959)
'Turbulence',
McGraw Hill, New York.
45. ILEGBUSI J. O. & SPALDING D. B. (1982a)
'A Steady-unsteady Visualization Technique for Wake-Flow
Studies',
CFDU, Imperial College, London, Report No. CFD/82/12.
46. ISHIKAWA N. & DAILY J. W. (1978)
'Observation of Flow Characteristics in a Model I.C. Engine
Cylinder',
SAE Paper 780230.
47. JAMES E. H. (1972)
'Investigations of Turbulent Flow in the Combustion Chamber of a
Spark Ignition Engine',
Ph. D. Thesis, Loughborough University of Technology.
48. JAMES E. H. & LUCAS G. G. (1975)
'Turbulent Flow in Spark Ignition Engine Combustion Chambers',
SAE Paper 750886.

49. KASTNER L. S. , WILLIAMS T.J. & WHITE J. B. (1963)
'Poppet Inlet Valve Characteristics and their Influence on the Induction Process',
Proc. I. Mech. E. , 178, 955.
50. LANCASTER D. R. (1976)
'Effect of Engine Variables on Turbulence in a Spark Ignition Engine',
SAE Paper 760159.
51. LAUNDER B. E. & SPALDING D. B. (1972b)
'Turbulence Models and their Application to the Prediction of Internal Flows',
Heat and Fluid Flow Vol 2.
52. LAUNDER B. E. & SPALDING D. B. (1972a)
'Mathematical Models of Turbulence',
Academic Press. London.
53. MATTHEWS L. & WHITELAW J. H. (1971)
'The Prediction of Film Cooling in the Presence of Recirculating Flows with a Two-equation Turbulence Model',
Imperial College, London, Mech. Eng. Dept. Report HTS/71/31.
54. MAXWELL T. T. (1977)
'Numerical Modelling of Free-surface Flows',
Ph.D Thesis, University of London.

55. MAXWORTHY T. (1972)
'The Structure and Stability of Vortex Rings',
J. Fluid Mech., Vol. 51, pp.15-32.
56. MCCLOY D. & MCGUIGAN R. H. (1964)
'Some Static and Dynamic Characteristics of Poppet Valves',
Proc. Instn. Mech. Engrs. 1964-1965.
57. MERZKIRCH W. (1974)
'Flow Visualization',
Academic Press, Inc.
58. MONAGHAN M. L. & PETTIFER H. F. (1981)
'Air Motion and Its Effect on Diesel Performance and Emissions',
SAE Paper 810255.
59. MORTON W. D. (1970)
'Match the Engine Intake System to the Application and Environment',
SAE Paper 700533.
60. MURTHY B. S. & GAJENDRA BABU M. K. (1976)
'Simulation and Evaluation of Exhaust and Intake System of a Four-Stroke Spark Ignition Engine',
SAE Paper 760763.

61. OHATA A. & ISHIDA Y. (1982)
'Dynamic Inlet Pressure and Volumetric Efficiency of Four Cycle Four Cylinder Engine'.
SAE Paper 820407.
62. QIN H.Q. (1983)
'Steady Flow Through a Seatless Poppet Valve Opening'.
CFDU PDR/3.
63. QIN H.Q. (1983)
'The Flow Characteristics of sudden Axisymmetric Expansion'.
CFDU PDR/14.
64. QIN H.Q. (1983)
'Steady Flow Through a Poppet Valve with Conical seat and the Effect of the Valve Geometry on the Discharge Coefficient'.
CFDU PDR/8.
65. QIN H.Q. & SPALDING D.B. (1984)
'Circumferential Pseudo-Diffusion In a Toroidal Vortex'.
PCH Vol. 5, No. 1, pp 85, 1984.
66. RICHARD D.R. (1984)
'Fundamentals of Reciprocating Engine Airflow Part 1: Valve Discharge and Combustion Chamber Effects'.
SAE Paper 840337.

67. RICHMAN R.M. & REYNOLDS W.C. (1984)
'The Development of a Transport Cylinder Engine for Piston Engine Fluid Mechanics Research',
SAE Paper 840379.
68. ROACHE P.J. (1976)
'Computational Fluid Dynamics',
Hermosa Publications.
69. ROBERT L.W. (1972)
'Turbulent Swirling Flows with Recirculation',
Ph.D. Thesis. University of London.
70. ROMOS J.I., HUMPHREY J.A.C. & SIRIGNANO W.A. (1979)
'Numerical Prediction of Axisymmetric Laminar and Turbulent Flows in Motored Reciprocating Internal Combustion Engines',
SAE Paper 790356.
71. ROMOS J.I. & SIRIGNANO W.A. (1980)
'Axisymmetric Flow Model in a Piston-Cylinder Arrangement with Detailed Analysis of the Valve Region',
SAE Paper 800286.
72. ROMAS J.I. & SIRIGNANO W.A. (1981)
'Turbulent Flow Field in Homogeneous - Charge Spark-Ignition Engines',
Proc. of Eighteenth International Combustion Symp. 1825.

73. ROMAS J.I., GANY A. & SIRIGNANO (1981)
'Study of Turbulence in a Motored Four-Stroke Internal Combustion Engine',
AIAA Journal Vol. 19, No. 5 1981 p. 595.
74. ROSTEN H.I. & SPALDING D.B.
'The Mathematical Basis of the PHOENICS-EARTH Computer Code',
CHAM TR/586.
75. SCHLICHTING H. (1968)
'Boundary Layer Theory',
McGraw-Hill, New York.
76. SPALDING D.B. (1969)
'The Prediction of Two-dimensional, Steady Turbulent Flows',
Imperial College, London, Heat Transfer Section Report
EF/TN/A/16.
77. SPALDING D.B. (1972b)
'A Two-equation Model of Turbulence',
VDI-Forsch Heft 549.
78. SPALDING D.B. (1976)
'Turbulence Models',
HTS/76/17.

79. SPALDING D. B. (1977)
'Diesel-Combustion Modelling Present Status and Future Prospects'.
HTS/77/28.
80. SPALDING D. B. (1982b)
'Four lectures on the PHOENICS Computer Code'.
CFDU/82/5
81. SPALDING D. B. (1981)
'A General Purpose Computer Program for Multi-dimensional One and Two-phase Flow'.
Presented at Lehigh IMACS Conference, July; published in
Mathematics and Computer in Simulation XIII 1981 pp. 267-276.
82. SPALDING D. B. (1982)
'Representation of Combustion in Computer Models of Spark-Ignition'.
83. SPALDING D. B. (1980)
'Numerical Computation of Flows of Two Phases Separated by a Moving Interface'.
HTS/80/4.
84. SPALDING D. B. (1983)
'Towards a Two-fluid Model of Turbulent Combustion in Gases, with Special Reference to the Spark Ignition Engine'.
IMechE 1983 C53/83.

85. TABACZYNSKI R.J. (1982)
'Effect of Inlet and Exhaust System Design on Engine Performance',
Ford Motor Co.
86. TABACZYNSKI R.J. DAVIS G.C. & BELAIRE R.C. (1984)
'The Effect of Intake Valve lift on Turbulence Intensity and Burnrate in S.I. Engine-Model Versus Experiment',
SAE Paper 840030.
87. TANABE Seichi, OHIGASHI Shunichi & HAMAMOTO Yoshisuke (1969)
'Gas Flow Velocity in Inlet Pipe, Exhaust Pipe and cylinder of Two-Stroke Cycle Engine',
Bulletin of the JSME 621.434.06.
88. TANAKA K. (1929)
'Air Flow Through Suction Valve of Conical Seat',
Aeronautical Research Institute Report, Tokyo Imperial University,
part 1, 260; part 2.
89. TAYLOR M.K.P. & WHITELAW J.H. (1984)
'Velocity Characteristics in the Turbulent Near Wakes of Confined Axisymmetric Bluff Bodies',
J. Fluid Mech. (1984), Vol. 139, pp.391-416.

90. TAYLOR C. F. (1980)
'The Internal Combustion Engine in Theory and Practice',
Vol. 1, M.I.T. Press, M.I.T., Cambridge, Massachusetts,
Fourth Printing, 1980.
91. TENEKES H. & LUMLEY J. L. (1972)
'A First Course in Turbulence',
The MIT Press.
92. TINDAL M. J., WILLIAMS T. J. & ALDOORY M. (1982)
'The Effect of Inlet Port Design on Cylinder Gas Motion in Direct
Injection Diesel Engines',
Proc. Symp. on Flows in Internal Combustion Engines, Arizona.
93. TOWNSEND A. A. (1961)
'Equilibrium Layers and Wall Turbulence',
J. Fluid Mech., 48, part 3, pp. 477-505.
94. VAN DRIEST E. R. (1956)
'On Turbulent Flow near Wall',
J. Aero Sci 23, pp. 1007.
95. WATKINS A. P. (1977)
'Flow and Heat Transfer in Piston/Cylinder Assemblies',
Ph.D. Thesis, University of London.

96. WATANABE E. & FUKUTANI I. (1979)
'An Analysis of the Volumetric Efficiency Characteristics of 4-Stroke Cycle Engines Using the Mean Inlet Mach Number M_{im} '.
SAE Paper 790484.
97. WHITE F.M. (1974)
'Viscous Fluid Flow'.
Press. U.S.A.
98. WIGLEY G., HAWKINS M.G. (1978)
'Three Dimensional Velocity Measurements by Laser Anemometry in a Diesel Engine Cylinder Under Steady State Inlet Flow Conditions'.
SAE PAPER 780060.
99. WILLIAMS T.J. & COX J.B. (1975)
'The Influence of Exhaust Poppet Valve Gas Flow on Reciprocating Engine Noise'.
Proc. Instn. Mech. Engrs. Vol 189 45/75.
100. WITZE P.O. (1980b)
'Influence of Air Motion Variation on the Performance of a Direct-Injection Stratified-Charge Engine'.
I. Mech. E. Paper C394/80.

101. WITZE P. O. (1977)
'Hot-Wire Measurements of the Turbulence Structure in a Motored Spark-Ignition Engine'.
SAND 77-8233 AIAA 14th Aerospace Sciences Monthly.
102. WITZE P. O. (1977)
'Measurements of the Spatial Distribution and Engine Speed Dependence of Turbulent Air Motion in an I.C. Engine'.
SAE paper 770220.
103. WOODS W. , GARG R & BENSON R. S. (1963)
'Unsteady Flow in Pipes with Gradual or Sudden Area Changes'.
Proc. Inst. of Mech. Engrs. Vol. 78, 1963-1964.
104. WOOD G. B. , HUNTER D. U. , TAYLOR E. S. & TAYLOR C. F.
(1902)
'Air Flow Through Intake Valves'.
Transaction of SAE. 50, 212.
105. WOOD G. B. & KHAN S. R. (1965)
'An Experimental Study of Flow Through Poppet Valves'.
Proc. I. Mech. E. , 180, 32.
106. WOOD W. A. & KHAN S. R. (1965)
'Experimental Study of Flow Through Poppet Valves'.
Proc. Instn. Mech. Engrs. 1965-1966.

107. WOOD W.A. (1967)
'Steady Flow Tests on Twin Poppet Valves'.
Proc. Instn. Mech. Engrs. 1967-1968.
108. PEDLOSKY, J. (1979)
Geophysical Fluid Dynamics. Springer.

NOMENCLATURE

<u>Symbol</u>	<u>Meaning</u>	<u>Location of first Appearance</u>
A_1	Peripheral lift area for a seatless valve	Ean A-1
A_{15}	Peripheral lift area for a valve with seat angle	Ean A-2
A_e	Effective area	Ean A-3
C	Convection flux (In chapter 3)	Ean 3.2-6
C_D	Discharge coefficient	Fig. 5.6-9
C_μ	Constant for turbulence equation	Ean 2.3-4
D	Diffusion flux (In chapter 3) Diameter of the cylinder	Fig. 4.3-1
D_0	Diameter of the small pipe	Fig. 4.3-1
D_s	Diameter of the valve stem	Fig. 8.21-1

E	Constant in log-law of the wall	Ean 2.5-1
G_R	Generation of turbulence energy	Ean 2.3-8
g	Gravitational acceleration	Chapter 8
J_ϕ	Flux of ϕ	Ean 3.2-1
k	Turbulence kinetic energy	Ean 2.3-1
L	Length	Flg. 4.3-1
L_v	Valve Lift	Flg. 1.1-2
$\dot{m}$	Mass transfer rate per unit area through surface	Ean 3.4-1
P	Mean pressure	Ean 2.2-4
$\bar{p}$	Instantaneous pressure	Ean 2.2-4
p'	Pressure fluctuation	Ean 2.2-4
R_1	Valve-stem radius	Flg. 8.24-1
R_2	Small pipe radius	Flg. 8.24-1
R_f	Under-relaxation factor	Ean 3.4-5
S_ϕ	Source of the equation for ϕ	Ean 2.2-9

S_K	Source for the κ equation	Ean 2.3-6
S_ϵ	Source for the ϵ equation	Ean 2.3-7
S_p	Implicit part of a linearised source	Ean 3.2-11
S_u	Explicit part of a linearised source	Ean 3.2-11
t_0	Initial time	Chapter 7
t	Time (seconds)	Chapter 7
U	Mean velocity component along the stream direction	Chapter 2
$\bar{u}_l$	Instantaneous velocity component along l'th direction	Chapter 2
u_l	Fluctuating velocity component along l'th direction	Chapter 2
u_τ	Friction velocity	Ean 2.5-1
$\overline{u_i u_j}$	Reynolds stress tensor	Ean 2.2-6
$\overline{u_l' \phi'}$	Turbulent flux of ϕ	Ean 2.2-9

V	Mean-velocity component along the cross-stream direction	Chapter 2
W	Mean-velocity component along the axial direction	Chapter 8
X	Coordinate along main-flow direction	Chapter 2
x_i	Generalized cartesian tensor coordinate	Ean 2.2-1
y	Coordinate cross the main-flow direction	Chapter 2
y^+	Non-dimensional distance from	Ean 2.5-3
Z	Coordinate along axial direction	Chapter 8

Greek Symbols

δ_{ij}	kronecker delta	Ean 2.3-1
ϵ	Rate of dissipation of turbulence energy	
Γ_κ	Exchange coefficient for κ	Ean 2.3-6
Γ_ϵ	Exchange coefficient for ϵ	Ean 2.3-7
μ_l	Laminar viscosity	Ean 2.3-5
μ_t	Turbulent viscosity	Ean 2.3-5
μ_{eff}	Effective viscosity	Ean 2.3-5
k	Von-Karman constant in the logarithmic law	Ean 2.5-1
η	Composite coordinate	Appendix B
ζ	Composite coordinate	Appendix B
ϕ	Time-mean value of a scalar quantity	Ean 2.2-8
$\bar{\phi}$	Instantaneous value of a scalar quantity	Ean 2.2-8

ϕ'	Fluctuating component of a scalar quantity	Ean 2.2-8
τ_w	Wall shear stress	Ean 2.5-6
ρ	Density	Ean 2.2-1
λ	Relaxation factor	Ean 3.4-3

Subscripts

E. W. N.S.P	In Chapter 3, for east, west, north, south and central nodes respectively of the finite- domain grid	Chapter 3
e.w.n.s	East, west, north and south interfaces of the control volume respectively	Chapter 3
k	Turbulence kinetic energy	Chapter 2
ϵ	Dissipation rate of turbulence kinetic energy	Chapter 2
w	wall	

APPENDIX A

Discharge Coefficient

The discharge coefficient, C_D , is defined as the ratio of effective flow area, A_e , to some geometric area representative of the valve configuration. The effective area is considered here to be the throat area of an imaginary, frictionless, parallel-exit nozzle and through which the measured flow passes. The choice of the geometric area is arbitrary and various reasonable choices exist. Most widely used areas are minimum area (Kastner et al. 1963) and peripheral lift area A_1 (Annand, 1969). The latter is chosen for the present study as follows:

$$A_1 = \pi D_v L_v \quad .(A-1)$$

$$A_{15} = \pi [(D_v - 2w)L_v] \quad .(A-2)$$

the first is for a seatless valve, and the latter for a valve with a seat angle.

$$\text{Hence, } C_D = \frac{A_e}{A_1} \quad .(A-3)$$

$$\text{and } A_e = \frac{\dot{m}}{\sqrt{2\rho|\Delta p|}} \quad .(A-4)$$

where Δp is the stagnation pressure drop across the valve;

D_v is the valve diameter ;

L_v is the valve lift ;

$\dot{m}$ is the mass flow rate .

APPENDIX B

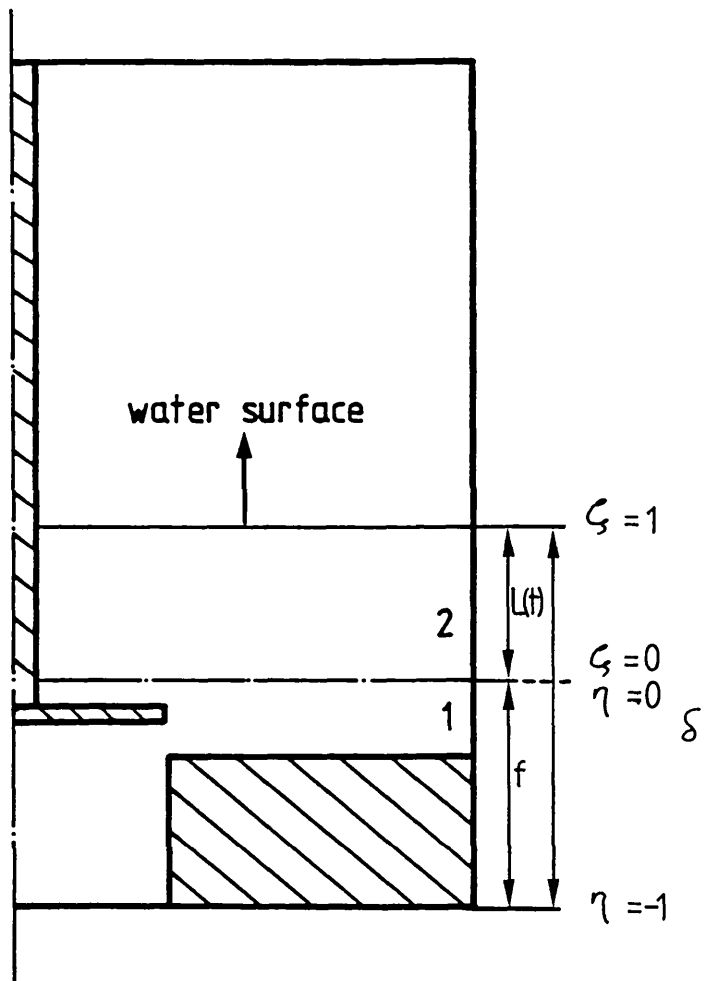
Transformation of Equations for a Composite Grid

The conservation equations in cylindrical-polar coordinates for laminar flow are:

$$\text{Continuity. } \frac{\partial \rho}{\partial t} + \frac{\partial (\rho w)}{\partial Z} + \frac{1}{r} \frac{\partial (\rho v r)}{\partial r} = 0 \quad .(B-1)$$

The transport equation for a general variable ϕ .

$$\frac{\partial}{\partial t} (\rho \phi) + \frac{\partial}{\partial Z} [\rho w \phi - \Gamma \frac{\partial \phi}{\partial Z}] + \frac{1}{r} \frac{\partial}{\partial r} [r (\rho v \phi - \Gamma \frac{\partial \phi}{\partial r})] = S_\phi \quad .(B-2)$$



- (1) fixed grid
- (2) moving grid

Figure B-1 A composite grid-system.

A composite grid consists of a "moving part", which stretches uniformly with the movement of the boundary, and a "fixed part", as shown in Figure B-1.

For the moving part, a transformation is required to replace the co-ordinate set (r, Z, t) by the set (r, ζ, t) , where the new axial-coordinate, ζ , is specified as

$$\zeta = \zeta (Z, t) \quad .(B-3)$$

A variable ϕ can be expressed in both coordinate systems, thus

$$\phi = \phi_1 (r, Z, t) = \phi_2 (r, \zeta, t), \quad (B-4)$$

Differentiation of (B-4) yields:

$$\frac{\partial \phi_1}{\partial r} dr + \frac{\partial \phi_1}{\partial Z} dZ + \frac{\partial \phi_1}{\partial t} dt = \frac{\partial \phi_2}{\partial r} dr + \frac{\partial \phi_2}{\partial \zeta} d\zeta + \frac{\partial \phi_2}{\partial t} dt \quad .(B-5)$$

$$\text{But as } d\zeta = \frac{\partial \zeta}{\partial Z} dZ + \frac{\partial \zeta}{\partial t} dt \quad .(B-6)$$

$$\begin{aligned} \text{thus, } \frac{\partial \phi_1}{\partial r} dr + \frac{\partial \phi_1}{\partial Z} dZ + \frac{\partial \phi_1}{\partial t} dt &= \frac{\partial \phi_2}{\partial r} dr + \frac{\partial \phi_2}{\partial \zeta} \frac{\partial \zeta}{\partial Z} dZ \\ &+ \frac{\partial \phi_2}{\partial \zeta} \frac{\partial \zeta}{\partial t} dt + \frac{\partial \phi_2}{\partial t} dt \quad .(B-7) \end{aligned}$$

So three coordinate transformation formulae are obtained as:

$$\frac{\partial \phi_1}{\partial r} = \frac{\partial \phi_2}{\partial r}$$

$$\frac{\partial \phi_1}{\partial Z} = \frac{\partial \phi_2}{\partial \zeta} \frac{\partial \zeta}{\partial Z}$$

$$\text{and } \frac{\partial \phi_1}{\partial t} = \frac{\partial \phi_2}{\partial t} + \frac{\partial \phi_2}{\partial \zeta} \frac{\partial \zeta}{\partial t} \quad , (B-8)$$

For the present study, $\zeta = \frac{Z}{L(t)}$, where $L(t)$ is the instantaneous

distance of water surface from the grid interface as shown in Figure B-1. Then planes at $\zeta = 0$ and $\zeta = 1$ correspond to the grid interface and the water surface respectively, and, grid lines in both ζ and Z spaces have the same relative spacings.

Further manipulation yields equations for the new coordinates system:

$$\begin{aligned} \frac{1}{L} \frac{\partial}{\partial t} (\rho L \phi) + \frac{1}{L} \frac{\partial}{\partial \zeta} (\rho w \phi) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho v \phi) - \frac{1}{L} \frac{\partial}{\partial \zeta} \left(\frac{\Gamma}{L} \frac{\partial \phi}{\partial \zeta} \right) - \frac{1}{r} \frac{\partial}{\partial r} \left(r \Gamma \frac{\partial \phi}{\partial r} \right) \\ = S \phi \end{aligned} \quad , (B-9)$$

As $L = s-f$

where s is the axial length of the whole calculation domain, and f is the axial length of the fixed grid domain. It can be obtained:

$$\begin{aligned} \frac{1}{s-f} \frac{\partial}{\partial t} [\rho \phi (s-f)] + \frac{1}{s-f} \frac{\partial}{\partial \zeta} (\rho w \phi) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho v \phi) - \frac{1}{s-f} \frac{\partial}{\partial \zeta} \left(\frac{\Gamma}{s-f} \frac{\partial \phi}{\partial \zeta} \right) \\ - \frac{1}{r} \frac{\partial}{\partial r} \left(r \Gamma \frac{\partial \phi}{\partial r} \right) = S \phi \end{aligned} \quad , (B-10)$$

where $\bar{w}$ is the relative axial velocity and

$$\bar{w} = w - \zeta \frac{dL}{dt} = w - \zeta w s \quad , (B-11)$$

$$\text{with } w_S = \zeta \frac{dL}{dt} \quad .(B-12)$$

In the fixed part, a nondimensional axial coordinate $\eta = \frac{Z}{f}$

may be defined where $-f \leq Z \leq 0$, such that the inlet plane is located at:

$\eta = -1$ and the grid-interface at $\eta = 0$.

By substituting (B-11) into B-2) the latter becomes:

$$\begin{aligned} \frac{1}{f} \frac{\partial}{\partial t} (\rho f \phi) + \frac{1}{f} \frac{\partial}{\partial \zeta} (\rho \bar{w} \phi) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho v \phi) - \frac{1}{f} \frac{\partial}{\partial \zeta} \left(\frac{r}{f} \frac{\partial \phi}{\partial \zeta} \right) \\ - \frac{1}{r} \frac{\partial}{\partial r} \left(r \Gamma \frac{\partial \phi}{\partial r} \right) = S \phi \end{aligned} \quad .(B-13)$$

Obviously equations (B-10) and (B-13) are similar and both can be written as:

$$\frac{1}{X} \frac{\partial}{\partial t} [\rho \phi X] + \frac{1}{r} \frac{\partial}{\partial r} [r (\rho v \phi - \Gamma \frac{\partial \phi}{\partial r})] + \frac{1}{X} \frac{\partial}{\partial \zeta} [\rho \bar{w} \phi - \frac{r}{X} \frac{\partial \phi}{\partial \zeta}] = S \phi \quad .(B-14)$$

$$\text{where } X = \begin{cases} f & -f \leq Z \leq 0 \\ \delta - f & 0 \leq Z \leq \delta - f \end{cases} ;$$

$$\bar{w} = \begin{cases} w & -f \leq Z \leq 0 \\ w - \zeta w_S & 0 \leq Z \leq \delta - f \end{cases} .$$

$$\text{or } \bar{w} = w - \zeta w_S$$

$$w_S = 0, \text{ for } -f \leq Z \leq 0 \text{ in the fixed part}$$

$$\text{and } w_S = \zeta \frac{dL}{dt}, \text{ for } 0 \leq Z \leq \delta - f \text{ in the moving part.}$$

APPENDIX C

Particle-Tracking

A particle-tracking technique was developed by Maxwell (1977) to investigate the behaviour of free surfaces in a variety of flow situations; it was later used by Awn (1979) for the calculation of flows with a moving interface. In the present study, the TRACK Code provided by Castrejon is modified by author to trace the vortex pattern downstream of the valve. Its principle is described below.

The Basic Idea of the Particle-tracking for an Interface

A series of imaginary particles are distributed, at a specified initial time, on the interface. Each of these particles is characterised by co-ordinates Z_p and Y_p , where p denotes the particle. These particles are moved at the end of a specified time step in accordance with the prevailing velocity distribution in the flow domain. The position of a particle is described by the following equations:

$$\frac{\partial Z_p}{\partial t} = U_p \quad .(C-1)$$

$$\text{and } \frac{\partial Y_p}{\partial t} = V_p \quad .(C-2)$$

They may be written in finite-difference forms as

$$Z_p = Z_{p0} + U_p \Delta t \quad .(C-3)$$

$$\text{and } Y_p = Y_{p0} + V_p \Delta t \quad .(C-4)$$

where Z_{p0} and Y_{p0} are the coordinates of the particle at time t_0 and U_p and V_p are the velocity components with which the particle moves through the finite difference mesh. From the new particle location after a time step, Δt , the new interface may be established.

There are several methods of calculating the particle velocities. In the present study, the chosen method consists of evaluating its velocity components by a direct linear interpolation of those of its neighbouring four particles. Special attention must be given to particles which lie near the boundaries, in which, for the present study, all velocities used for interpolation are contained in the fluid side.

Flow Pattern Tracking

For the present study, one particle is placed at the injection point of the tracer, at a short distance below the cylinder bottom. At the end of the particle's advancement, another particle is placed at the same injection position. Thus, over a number of particle advancement steps, a string of particles is available to indicate the instantaneous locus of all fluid particles which have passed through the injection point over a specified time interval. Along the locus, the instantaneous location of each particle is specified by the above equations.

Because of the irregular fluid motion, the spacing of the particles along the locus filament may become very uneven. In order to provide a locus of evenly distributed particles, checks are made of the spacing between adjacent particles. If it is larger than a prescribed maximum value, a new particle is inserted between them; and if it is smaller than a minimum distance, one of them is deleted.

APPENDIX D

CODING OF SATELLITE

SUBROUTINE SATLIT

```
C-----
$INCLUDE 9.CMNGUSSI.FTN/G
$INCLUDE 9.GUSSEQUI.FTN/G
$INCLUDE 9.CMNGRFIC.FTN/G
C-----
C---  GROUP 1. FLOW TYPE :
      CARTES=.FALSE.
C-----
C---  GROUP 2. TRANSIENCE :
      STEADY=.FALSE.
      ATIME=0.
      FSTEP=1
      LSTEP=360
      TLAST=180.
      TFRAC(1)=-1000.
      TFRAC(2)=0.005
C-----
C---  GROUP 4. Y-DIRECTION :
      NY=27
      YVLAST=0.0865
      YFRAC(1)=-10.
      YFRAC(2)=0.0289
      YFRAC(3)=4.
      YFRAC(4)=.00578
      YFRAC(5)=2.
      YFRAC(6)=.037
      YFRAC(7)=7.
      YFRAC(8)=.065
      YFRAC(9)=3.
      YFRAC(10)=.042
      YFRAC(11)=1.
      YFRAC(12)=.025
      NY1=10
      NY1P1=NY1+1
      NY1P=NY1+4
C-----
C---  GROUP 5. Z-DIRECTION :
      NZ=45
      ZFRAC(1)=-5.
      ZFRAC(2)=.006
      ZFRAC(3)=2.
      ZFRAC(4)=.0025
      ZFRAC(5)=4.
      ZFRAC(6)=.0005
      ZFRAC(7)=8.
      ZFRAC(8)=.001
      ZFRAC(9)=10.
      ZFRAC(10)=.0005
      ZFRAC(11)=2.
      ZFRAC(12)=.001
      ZFRAC(13)=3.
      ZFRAC(14)=.002
      ZFRAC(15)=3.
```

```
ZFRAC(16)=.001
ZFRAC(17)=4.
ZFRAC(18)=.0005
ZFRAC(19)=4.
ZFRAC(20)=.00025
NZ1=7
NZV=20
NZ1P1=NZ1+1
NZVM1=NZV-1
NZVP1=NZV+1
C-----
C--- GROUP 6. MOVING GRID :
      MGRID=2
      IZW1=22
      IZW2=NZ
      AZW2=7.683E-4
      BZW2=0.0
C-----
C--- GROUP 7. BLOCKAGE: BLOCK<.F.>.IPLANE.IPWRT
      BLOCK=.TRUE.
C-----
C--- GROUP 8. DEPENDENT VARIABLES TO BE SOLVED FOR OR STORED
      :
      SOLVAR(P1)=.TRUE.
      SOLVAR(PP)=.TRUE.
      SOLVAR(V1)=.TRUE.
      SOLVAR(W1)=.TRUE.
C-----
C--- GROUP 10 PROPERTIES:
      EMULAM=1.E-3
      EMU1=EMULAM
      RHO1=1.E3
C-----
C--- GROUP 13 INITIAL FIELDS :
      FIINIT(1)=10101.
      FIINIT(5)=10101.
      FIINIT(7)=10101.
C-----
C--- GROUP 15 TO 24: REGIONS 1 TO 10
C--- CYLINDER WALL
      CALL PLACE(2,NORTH,1,1,NY,NY,1,NZ)
      CALL COVAL(2,W1,WALL,0.0)
C--- STEP WALL
      CALL PLACE(4,NORTH,1,1,NY1,NY1,1,NZ1)
      CALL COVAL(4,W1,WALL,0.0)
C--- STEP WALL(VERTICAL)
      CALL PLACE(5,LOW,1,1,NY1P1,NY,NZ1P1,NZ1P1)
      CALL COVAL(5,V1,WALL,0.0)
C--- VALVE WALL
      CALL PLACE(6,HIGH,1,1,1,NY1,NZVM1,NZVM1)
      CALL COVAL(6,V1,WALL,0.0)
C--- VALVE WALL(BACK)
      CALL PLACE(7,LOW,1,1,1,NY1,NZVP1,NZVP1)
      CALL COVAL(7,V1,WALL,0.0)
C--- STEM WALL
      CALL PLACE(8,SOUTH,1,1,2,2,1,NZ)
      CALL COVAL(8,W1,WALL,0.0)
C-----
```

C--- GROUP 25 GROUND STATION :
GROSTA=. TRUE.

C-----

C--- GROUP 26 SOLUTION TYPE AND RELATED PARAMETERS :
WHOLEP=. TRUE.

C-----

C--- GROUP 27 SWEEP AND ITERATION NUMBERS :
FSWEEP=1
LSWEEP=300
LITER(2)=10
LITER(7)=10

C-----

C--- GROUP 28 TERMINATION CRITERIA :
ENDIT(2)=1. E-4
ENDIT(7)=1. E-3

C-----

C--- GROUP 29 RELAXATION :
DTFALS(5)=0.1
DTFALS(7)=0.1

C-----

C--- GROUP 33 MONITOR PRINT-OUT :
IYMON=NY1+1
IZMON=NZ1P1+1

C-----

APPENDIX E

CODING OF GROUND

```

      SUBROUTINE GROUND(IRN,ICHAP,ISTP,ISWP,IZED,INDVAR)
$INCLUDE 9.CMNGUSSI.FTN/G
$INCLUDE 9.GUSSEQUI.FTN/G
C-----
C   CHAPTER 0: ALWAYS ACCESSED WHEN GROUND CALLED.
C-----
      PARAMETER (NYM=27,NZM=45,NYNZ=1215)
      DIMENSION LSPDA(1),ISPDA(1),RSPDA(1)
      DIMENSION CPHI(NYM,1),VPHI(NYM,1),CM(NYM,1),VM(NYM,1)
                                                    DIMENSION
DENS(NYM,1),VOL(NYM,1),AREA(NYM,1),PRES(NYM,1)
      DIMENSION WINOM(NYM),PSIDD(NYM,NZM),AREAN(NYM,1)
      DIMENSION ZG(NZM),WINN(NYM),GV(NYM,1),GW(NYM,1)
      DIMENSION GP(NYM,1),AMSD(NYM),V1(NYM,1),W1(NYM,1)
                                                    DIMENSION
WT(NYM,1),DYG(NYM),R(NYM),PSID(NYM,1),RATW(NYM,1)
      DATA NDIM/210/
      DATA NY1,NPTUB1,NPTUB2,NPTUB3/10,4,16,20/
      DATA DO,DOO/0.05,0.005/
      DATA WIN,RHO/.0124,1.0E3/
      DATA WV,VALD,VALL/0.0,0.05,0.015/
C---  FOR SEATLESS VALVE
C
      WINOM/.9975,.9775,.9375,.885,.7975,.6975,.5775,.4375,
C   1      .2775,.0975,17*0./
C   DATA WREF/0.03375/
C---  FOR SEATED VALVE
C
      WINOM/0.,0.,.2345,.6094,.8594,.9843,.9843,.8594,.6094,
C   1      .2345,17*0./

      PAI=4.*ATAN(1.)
C-----
C   CHAPTER 1: CALLED AT THE START OF EACH TIME STEP.
C-----
      100 CONTINUE
      IPLANE=2
      IF(ISTP.GT.1) RETURN
      CALL DETGJ(IPLANE)
      RETURN
C-----
C   CHAPTER 2: CALLED AT THE START OF EACH SWEEP.
C-----
      200 CONTINUE
      AMASS=0.018
C
C---  FOR SEATLESS VALVE
C
C   AMASS=AMASS/(.25*PAI*DO*DO)
C   WREF=2.*AMASS/RHO
C
C---  FOR SEATED VALVE
C
C   AMASS=AMASS/(.25*PAI*(DO*DO-DOO*DOO))

```

```

      WREF=1.5*AMASS/RHO
C
C--- CALCULATE INLET VELOCITY
C
      DO 301 IY=1,NY
      WINN(IY)=WINOM(IY)*WREF
301 CONTINUE
      RETURN
C-----
C      CHAPTER 5: GROUND CALLED WHEN SOURCE TERM IS
COMPUTED.
C-----
      500 CONTINUE
C
C--- INLET BOUNDARY CONDITION
      IF(IZED.NE.1) RETURN
      DO 501 IY=1,NY1
      CM(IY,1)=1.E-10
501 VM(IY,1)=1.E10*AMASS
      IF(INDVAR.NE.9) GO TO 504
      CALL ADD(9,1,1,1,NY1,6.,CM,VM,CPhi,VPhi,NY,NX)
      GO TO 599
504 IF(INDVAR.EQ.7) GO TO 505
      GO TO 599
505 DO 502 IY=1,NY1
      CPhi(IY,1)=0.0
502 VPhi(IY,1)=WINN(IY)
      GO TO 510
510 CALL ADD(INDVAR,1,1,1,NY1,6.,CM,VM,CPhi,VPhi,NY,NX)
599 CONTINUE
      RETURN
C-----
C      CHAPTER 7: CALLED AT END OF EACH SLAB-WISE
CALCULATION.
C-----
      700 CONTINUE
      CALL GET(5,V1,NY,NX)
      CALL GET(7,W1,NY,NX)
      CALL GET(1,PRES,NY,NX)
      CALL GET1D(129,DYG,NY)
      CALL GET(105,AREA,NY,NX)
      CALL GET(104,AREAN,NY,NX)
      CALL GET(2,DENS,NY,NX)
C
C--- MAXIMUM VELOCITY AT EACH CROSS-SECTION
C
      LSWPM1=LSWEEP-1
      IF(ISWP.NE.LSWPM1) GO TO 703
      BIG=W1(1,1)
      BIGV=V1(1,1)
      DO 707 IY=1,NY
      IF(W1(IY,1).GT.BIG) BIG=W1(IY,1)
      IF(V1(IY,1).GT.BIGV) BIGV=V1(IY,1)
707 CONTINUE
C
C--- CALCULATION OF THE STREAM FUNCTION
C
      PSID(1,1)=0.0

```

```

      IYP1=IY+1
      DO 701 IY=1,NY
      IYM1=IY-1
      IF(IY.EQ.1) IYM1=AMAX0(1,IY-1)
701  PSID(IY,1)=DENS(IY,1)*W1(IY,1)*AREA(IY,1)+PSID(IYM1,1)
      DO 706 IY=1,NY
      IJ=NY-IY+1
706  PSID(IJ,IZED)=PSID(IY,1)
      CALL SET(20,1,NX,1,NY,PSID,NY,NX)
C
C----  NORMALIZED VELOCITY
C
      DO 704 IY=1,NY
704  RATW(IY,1)=W1(IY,1)/BIG
      CALL SET(17,1,NX,1,NY,RATW,NY,NX)
C
703  IF(ISWP.NE.LSWEEP) GO TO 710
C
C----  PREPARING PRESSRE FOR CD
C
      IF(IZED.NE.NPTUB1) GO TO 712
      PRES1=PRES(NY,1)
      SUM=0.0
      DO 708 IY=1,NY1
      WGY1=W1(IY,1)*DYG(IY)+SUM
      SUM=WGY1
708  CONTINUE
      WAV1=SUM/YVLAST
      WRITE(6,8008) IND
      IND=4
712  IF(IZED.NE.NPTUB2) GO TO 711
      PRES2=PRES(NY,1)
      SUM=0.0
      DO 709 IY=1,NY
      WGY2=W1(IY,1)*DYG(IY)+SUM
      SUM=WGY2
709  CONTINUE
      WAV2=SUM/YVLAST
      P1=PRES1+.5*RHO1*WAV1*WAV1
      P2=PRES2+.5*RHO1*WAV2*WAV2
      DPE=ABS(P1-P2)
      WRITE (6,8001) WAV1,WAV2,DP
711  IF(IZED.EQ.3) PRE1=PRES(10,1)
      IF(IZED.EQ.12) PRE2A=PRES(NY,1)
      IF(IZED.EQ.13) PRE2B=PRES(NY,1)
      IF(IZED.EQ.14) PRE2C=PRES(NY,1)
      IF(IZED.EQ.15) PRE2D=PRES(NY,1)
      IF(IZED.EQ.16) PRE2E=PRES(NY,1)

      PRE2MX=AMAX1(PRE2A,PRE2B,PRE2C,PRE2D,PRE2E)
      PRE2MN=AMIN1(PRE2A,PRE2B,PRE2C,PRE2D,PRE2E)
710  CONTINUE
      IF(IZED.EQ.1) PT1MX=PRES(10,1)
      PS1MX=PT1MX+.5*RHO*WIN*WIN
      IF(IZED.EQ.13) PT2=PRES(NY,1)
      IF(IZED.EQ.12) W12=W1(NY,1)
      PS2=PT2+.5*RHO*W12*W12
      DPS=ABS(PS1MX-PS2)

```

```
C      IF(IZED.EQ.3) PT12=PRE(10,1)
      PSVMX=PT12+.5*RHO*WIN*WIN
      PT2MN=PRE2MN
      PS2MN=PT2MN+.5*RHO*W12*W12
      DPS1=ABS(PSVMX-PS2MN)

C      DPS2=ABS(PS1MX-PS2MN)

C      IF(IZED.EQ.5) PT1MN=PRE(10,1)
      PS1MN=PT1MN+.5*RHO*WIN*WIN
      DPS3=ABS(PS1MN-PSVMX)

C      IF(ISWP.NE.LSWEEP) GO TO 801
      IF(IZED.NE.1) GO TO 801
      SUMMAS=0.
      DO 801 IY=2,10
      GINMAS=RHO*W1(IY,1)*AREA(IY,1)
      SUMMAS=SUMMAS+GINMAS
801  CONTINUE
      RETURN

C-----
C      CHAPTER 8: CALLED AT THE END OF EACH SWEEP;
C-----

      800  CONTINUE
      IF(MOD(ISTP,NTPRIN).NE.0) RETURN

C
C----  WRITE STREAM FUNCTION
C
      WRITE(6,8004)
      WRITE(6,8001) ((PSIDD(IY,IX),IX=1,10),IY=1,NY)
      WRITE(6,8004)
8004  FORMAT(35X,30H-----PSID-----)
      WRITE(6,8001) ((PSIDD(IY,IX),IX=11,20),IY=1,NY)
      WRITE(6,8004)
      WRITE(6,8001) ((PSIDD(IY,IX),IX=21,30),IY=1,NY)
      WRITE(6,8004)
      WRITE(6,8001) ((PSIDD(IY,IX),IX=31,40),IY=1,NY)
      WRITE(6,8004)
      WRITE(6,8003) ((PSIDD(IY,IX),IX=41,NZM),IY=1,NY)
8003  FORMAT(1P5E12.3)

C
C----  CALCULATE CD
C
      PAI=4.*ATAN(1.)
      DP=ABS(PRE1-PRE2MN)
      AE=SUMMAS/SQRT(2.*RHO1*DPS2)
      CD2=2.*AE/((VALD-2.*WV)*VALL)

C
      AE1=SUMMAS/SQRT(2.*RHO1*DPS)
      CDB=2.*AE1/((VALD-2.*WV)*VALL)
      ACD=WIN*VALD*VALD/(4.*((VALD-2.*WV)*VALL))
      BCD=RHO/(2.*DPS)
      CDA=ACD*SQRT(BCD)

C
      BCD1=RHO/(2.*DPS1)
      AEV=SUMMAS/SQRT(2.*RHO1*DPS1)
      CDA1=2.*AEV/((VALD-2.*WV)*VALL)
```

C

BCD2=RHO/(2.*DPS2)
CDA2=ACD*SQRT(BCD2)
WRITE(6,8002)

8002

FORMAT(10X,3HACD,9X,3HBCD,10X,2HDP,9X,3HDPS,8X,4HMASS,
1 9X,2HAE,10X,2HCD,10X,3HCDA)
WRITE(6,8001) ACD,BCD,DPS2,DPS,SUMMAS,AE,CD2,CDA
WRITE(6,8001) AE1,CDB,CDA1,CDA2

C

8008 FORMAT(10X,I4)
8001 FORMAT(1P10E12.3)
RETURN

C-----

C CHAPTER 9: CALLED AT THE END OF EACH TIME STEP;
C NOT ACCESSED IF PARABOLIC.

C-----

900 CONTINUE
IF(ISTP.GT.90) GO TO 902
IF(ISTP.GT.150) GO TO 903
IF(ISTP.GT.280) GO TO 904
CALL TRACKH(ISTP,IPLANE,1.2E-3,2.3E-3)
RETURN
902 CALL TRACKH(ISTP,IPLANE,1.0E-3,2.0E-3)
RETURN
903 CALL TRACKH(ISTP,IPLANE,1.4E-3,2.6E-3)
RETURN
904 CALL TRACKH(ISTP,IPLANE,1.6E-3,3.0E-3)
RETURN

APPENDIX F

CODING OF TRACK

```

      SUBROUTINE TRACKH(ISTP,IPLANE,DISMIN,DISMAX)
C-----
C
C      SUBROUTINE FOR TRACING DYE FILAMENT LINE.
C-----
C
$INCLUDE 9,CMNGUSSI.FTN/G
$INCLUDE 9,GUSSEQUI.FTN/G
      COMMON/SET1/IXFA(10),IXLA(10),IYFA(10),IYLA(10),TRA(10),
1      INW(10),INE(10),INN(10),INS(10),IRP1
      DIMENSION XP(1200),YP(1200)
      DIMENSION U(27,45),V(27,45),XG(45),XU(45),YG(27),YV(27)
      DIMENSION W1(45,1),V1(45,1)
      DIMENSION XPLOT(1200),YPLOT(1200,1),YAXIS(1)
      LOGICAL TESTIN
      TESTIN=.FALSE.
      IF(.NOT.TESTIN) GO TO 9010
      WRITE(12,9020) IRP1
9020  FORMAT(1X,"IRP1 = ",1X,I2,/)
      WRITE(12,9030)
9030  FORMAT(1X,"INFORMATION FROM DETREG :",/)
      DO 9031 IA=1,IRP1
        WRITE(12,9032) IA,IXFA(IA),IXLA(IA),IYFA(IA),IYLA(IA),
1        TRA(IA),INW(IA),INE(IA),INN(IA),INS(IA)
9031  CONTINUE
9032  FORMAT(1X,5(I2,1X),1PE10.3,1X,4(I2,1X),/)
9010  CONTINUE
C
C..... RETRIEVE THE REQUIRED VARIABLES.
C
      NXSTO=NX
      IF(IPLANE.EQ.2) GO TO 300
      CALL GET(3,U,NY,NX)
      CALL GET(5,V,NY,NX)
      CALL GET1D(122,XG,NX)
      CALL GET1D(123,XU,NX)
      GO TO 310
C
300  DO 320 IZED=1,NZ
      CALL READIZ(IZED)
      CALL GET(7,W1,NY,1)
      CALL GET(5,V1,NY,1)
      DO 321 IY=1,NY
        U(IY,IZED)=W1(IY,1)
        V(IY,IZED)=V1(IY,1)
321  CONTINUE
320  CONTINUE
C
      CALL GET1D(132,XG,NZ)
      CALL GET1D(133,XU,NZ)
      NX=NZ
310  CALL GET1D(126,YG,NY)
```

```

      CALL GET1D(127,YV,NY)
C
C*****
C..... USER INSERTS HERE SIZE OF THE TIME STEP.
      DELTAT=0.2
      IF(ISTP.GT.40) DELTAT=0.5
C*****
      IF(ISTP.NE.1) GO TO 1
C*****
C..... USER INSERTS HERE PARTICLE DISTRIBUTION AT TIME=0.0
C      NOTE: IF IPLANE=2, XP MUST BE SET AS THE SET
COORDINATE.
C
C      INITIAL DISTRIBUTION FOR KASE 1.....
C
      YF=1.625E-2
      NPMAX=1
      DO 2 IP=1,NPMAX
        YP(IP)=YF*IP
      2 CONTINUE
      DO 3 IP=1,NPMAX
        XP(IP)=3.2E-2
      3 CONTINUE
C
C---      FOR RESTART
C      NPMAX=357
C      READ(23,999) (YP(IA),IA=1,NPMAX)
C      READ(23,999) (XP(IA),IA=1,NPMAX)
C
C      USER SECTION ENDS HERE.....
C
C*****
      WRITE(12,4)
      4 FORMAT(/,5X,"INITIAL PARTICLE DISTRIBUTION :",/)
      DO 5 IP=1,NPMAX
        WRITE(12,801) IP,XP(IP),YP(IP)
      5 CONTINUE
C
      1 CONTINUE
C*****
C
C      CHAPTER 0.- ADD OR DELETE PARTICLES AS NECESSARY
ACCORDING
C      TO THE VALUES OF DISMIN AND DISMAX
C
      IF(DISMIN.EQ.0.0) GO TO 95
      IP=1
      93 IF(IP.GE.NPMAX) GO TO 95
      IP=IP+1
      IPM1=IP-1
      IF(CARTES) GO TO 90
      IF(IPLANE.EQ.2) GO TO 90
      DIST=SQRT(((YP(IP)+RINNER)*SIN(XP(IP))-(YP(IPM1)+RINNER)*
      1 SIN(XP(IPM1)))**2.+((YP(IP)+RINNER)*COS(XP(IP))-
      2 (YP(IPM1)+RINNER)*COS(XP(IPM1)))**2.))
      GO TO 91
      90 DIST=SQRT((YP(IP)-YP(IPM1))**2.+(XP(IP)-XP(IPM1))**2.)
      IF(.NOT.TESTIN) GO TO 2958

```

```

WRITE(12,2959) DIST,IP,IPM1
2959 FORMAT(5X,"DIST(IP,IPM1)= ",1PE10.3,1X,12,1X,12,/)
2958 CONTINUE
91 IF(DIST.LT.DISMIN) GO TO 92
IF(DIST.LT.DISMAX) GO TO 93
NPMAX=NPMAX+1
NPMAX1=NPMAX-1
IF(.NOT.TESTIN) GO TO 2113
WRITE(12,1999) IP
1999 FORMAT(1X,"ADD PARTICLE WHEN IP ",1X,12)
2113 CONTINUE
DO 11 INS1=NPMAX1,IP,-1
INS1P1=INS1+1
XP(INS1P1)=XP(INS1)
YP(INS1P1)=YP(INS1)
11 CONTINUE
XP(IP)=0.5*(XP(IP)+XP(IPM1))
YP(IP)=0.5*(YP(IP)+YP(IPM1))
IF(.NOT.TESTIN) GO TO 2114
WRITE(12,2112)(YP(IA),IA=1,NPMAX)
WRITE(12,2111) NPMAX
2114 CONTINUE
GO TO 93
92 NPMAX=NPMAX-1
NPMAX1=NPMAX-1
IF(.NOT.TESTIN) GO TO 2115
WRITE(12,1998) IP
1998 FORMAT(1X,"DELETE PARTICLE WHEN IP ",1X,12)
2115 CONTINUE
DO 12 IDEL=IP,NPMAX
IDELP1=IDEL+1
XP(IDEL)=XP(IDELP1)
YP(IDEL)=YP(IDELP1)
12 CONTINUE
IF(.NOT.TESTIN) GO TO 2116
WRITE(12,2111) NPMAX
WRITE(12,2112)(YP(IA),IA=1,NPMAX)
WRITE(12,2112)(XP(IA),IA=1,NPMAX)
2112 FORMAT(1X,"NEW PARTICLE DISTRIBUTION ",/,12(1PE10.3,1X))
2116 CONTINUE
GO TO 93
2111 FORMAT(1X,"NPMAX =",12,/)
95 CONTINUE
C
C--- INJECTION OF PARTICLE AT EACH TIME STEP
IP=1
Y1=1.625E-2
Z1=3.2E-2
DIST=SQRT((YP(IP)-Y1)**2.+(XP(IP)-Z1)**2.)
IF(DIST.LT.1.E-3) GO TO 96
NPMAX=NPMAX+1
DO 97 IP1=NPMAX,2,-1
IP1M1=IP1-1
XP(IP1)=XP(IP1M1)
97 YP(IP1)=YP(IP1M1)
XP(IP)=Z1
YP(IP)=Y1
96 CONTINUE

```

```
C      IF(.NOT.TESTIN) GO TO 9040
      WRITE(12,9041)
9041  FORMAT(1X,"INITIAL PARTICLE DISTRIBUTION",/)
      WRITE(12,9042)(XP(IP),IP=1,NPMAX)
      WRITE(12,9042)(YP(IP),IP=1,NPMAX)
9042  FORMAT(1X,10(1PE10.3,1X))
9040  CONTINUE
C
C.....OPEN MAIN-DO-LOOP.
C
      IF(ISTP.LT.20) GO TO 111
      DO 1000 IDET=1,20
      DELTAT=.025
111  DO 1000 IP=1,NPMAX
C
C      CHAPTER 1.- SITUATING THE PARTICLE INSIDE THE DOMAIN.
C
      DO 100 IX=1,NX
      IF(XP(IP).LT.XU(IX)) GO TO 110
100  CONTINUE
110  DO 120 IY=1,NY
      IF(YP(IP).LT.YV(IY)) GO TO 130
120  CONTINUE
130  IXM1=IX-1
      IYM1=IY-1
      IXP1=IX+1
      IYP1=IY+1
      IZERO=0
C
C      IF(.NOT.TESTIN) GO TO 9050
      WRITE(12,9051) IP,IX,IY
9051  FORMAT(1X,"IP, IX, IY :",1X,3(12,2X),/)
9050  CONTINUE
C
C      CHAPTER 2.- DETERMINING WHETHER SPECIAL ACTIONS ARE
C      REQUIRED.
C
      ISPCAX=0
      ISPCAY=0
      IN=0
      IS=0
      IE=0
      IW=0
      DO 200 I=1,IRP1
      IF(IX.NE.IXFA(I).AND.IX.NE.IXLA(I)) GO TO 220
      IF(IYFA(I).EQ.IYLA(I)) GO TO 220
      ISPCAY=1
      IF(TRA(I).EQ.1) GO TO 230
      IW=1
      IF(INW(I).EQ.1) IW=2
      GO TO 200
230  IE=1
      IF(INE(I).EQ.1) IE=2
      GO TO 200
220  IF(IY.NE.IYFA(I).AND.IY.NE.IYLA(I)) GO TO 200
      IF(IXFA(I).EQ.IXLA(I)) GO TO 200
      ISPCAX=1
```

```
      IF(TRA(I).EQ.3) GO TO 240
      IS=1
      IF(INS(I).EQ.1) IS=2
      GO TO 200
240  IN=1
      IF(INN(I).EQ.1) IN=2
200  CONTINUE
C
      IF(.NOT.TESTIN) GO TO 9060
      WRITE(12,9061) ISPCAX,ISPCAY,IN,IS,IE,IW
9061  FORMAT(1X,"ISPCAX,ISPCAY,IN,IS,IE,IW :",1X,6(I2,1X),/)
9060  CONTINUE
C
C      CHAPTER 3. - ASSIGNING INDICES FOR NON-SPECIAL CASES.
C
      ILIFTX=0
      ILIFTY=0
      IF(XP(IP).GE.XG(IX)) ILIFTX=1
      IF(YP(IP).GE.YG(IY)) ILIFTY=1
C
C... INDICES FOR X (OR Z)-DIRECTION VELOCITIES.
      IUN=IY+ILIFTY
      IUS=IYM1+ILIFTY
      IUW=IXM1
      IUE=IX
C
C... INDICES FOR Y-DIRECTION VELOCITIES.
      IVN=IY
      IVS=IYM1
      IVW=IXM1+ILIFTX
      IVE=IX+ILIFTX
C
C... INDICES FOR X (OR Z)-DISTANCES.
      IXUW=IXM1
      IXUE=IX
      IXGW=IXM1+ILIFTX
      IXGE=IX+ILIFTX
C
C... INDICES FOR Y-DISTANCES.
      IYVN=IY
      IYVS=IYM1
      IYGN=IY+ILIFTY
      IYGS=IYM1+ILIFTY
C
C      CHAPTER 4. - MODIFYING INDICES FOR SPECIAL CASES.
C
      IF(ISPCAX.EQ.0.AND.ISPCAY.EQ.0) GO TO 400
      IF(IW.EQ.0) GO TO 410
      IUW=IZERO
      IF(ILIFTX.EQ.0) IVW=IZERO
      IF(IW.EQ.2) GO TO 420
      IF(ILIFTX.EQ.0) IVW=IVE
410  IF(IE.EQ.0) GO TO 420
      IUE=IZERO
      IF(ILIFTX.EQ.1) IVE=IZERO
      IF(IE.EQ.2) GO TO 420
      IF(ILIFTX.EQ.1) IVE=IVW
420  IF(IS.EQ.0) GO TO 430
```

```
      IVS=IZERO
      IF(ILIFTY.EQ.0) IUS=IZERO
      IF(IS.EQ.2) GO TO 430
      IF(ILIFTY.EQ.0) IUS=IUN
430  IF(IN.EQ.0) GO TO 400
      IVN=IZERO
      IF(ILIFTY.EQ.1) IUN=IZERO
      IF(IN.EQ.2) GO TO 400
      IF(ILIFTY.EQ.1) IUN=IUS
400  CONTINUE
C
C      CHAPTER 5. - ASSIGNING THE REQUIRED VARIABLES.
C
C... U (OR W)-VELOCITIES
      UNW=0.0
      IF(IUN.NE.IZERO.AND.IUW.NE.IZERO) UNW=U(IUN,IUW)
      USW=0.0
      IF(IUS.NE.IZERO.AND.IUW.NE.IZERO) USW=U(IUS,IUW)
      UNE=0.0
      IF(IUN.NE.IZERO.AND.IUE.NE.IZERO) UNE=U(IUN,IUE)
      USE=0.0
      IF(IUS.NE.IZERO.AND.IUE.NE.IZERO) USE=U(IUS,IUE)
C
      IF(.NOT.TESTIN) GO TO 9070
      WRITE(12,9071) UNW,USW,UNE,USE
9071  FORMAT(1X,"UNW,USW,UNE,USE :",1X,4(1PE10.3,2X),/)
9070  CONTINUE
C
C... V-VELOCITIES
      VNW=0.0
      IF(IVN.NE.IZERO.AND.IVW.NE.IZERO) VNW=V(IVN,IVW)
      VSW=0.0
      IF(IVS.NE.IZERO.AND.IVW.NE.IZERO) VSW=V(IVS,IVW)
      VNE=0.0
      IF(IVN.NE.IZERO.AND.IVE.NE.IZERO) VNE=V(IVN,IVE)
      VSE=0.0
      IF(IVS.NE.IZERO.AND.IVE.NE.IZERO) VSE=V(IVS,IVE)
C
      IF(.NOT.TESTIN) GO TO 9080
      WRITE(12,9081) VNW,VSW,VNE,VSE
9081  FORMAT(1X,"VNW,VSW,VNE,VSE :",1X,4(1PE10.3,2X),/)
9080  CONTINUE
C
C... X (OR Z)-DISTANCES.
      XUE=XU(IXUE)
      IF(IX.NE.1) GO TO 500
      XUW=0.0
      XGE=XG(IXGE)
      XGW=0.0
      IF(ILIFTX.EQ.1) XGW=XG(IXGW)
      IF(IW.EQ.2.AND.ILIFTX.EQ.0) XGW=-XG(IX)
      GO TO 510
500  IF(IX.NE.NX) GO TO 520
      XUW=XU(IXUW)
      XGW=XG(IXGW)
      XGE=XU(IX)+(XU(IX)-XG(IX))
      IF(IE.NE.2) XGE=XU(IX)
      GO TO 510
```

```
520 XUW=XU(IXUW)
    XGE=XG(IXGE)
    XGW=XG(IXGW)
510 CONTINUE
C
    IF(.NOT.TESTIN) GO TO 9090
    WRITE(12,9091) XUE,XUW,XGE,XGW
9091 FORMAT(1X,"XUE,XUW,XGE,XGW :",1X,4(1PE10.3,2X),/)
9090 CONTINUE
C
C... Y-DISTANCES.
    YVN=YV(IYVN)
    IF(IY.NE.1) GO TO 530
    YVS=0.0
    YGN=YG(IYGN)
    YGS=0.0
    IF(ILIFTY.EQ.1) YGS=YG(IYGS)
    IF(IS.EQ.2.AND.ILIFTY.EQ.0) YGS=-YG(IY)
    GO TO 550
530 IF(IY.NE.NY) GO TO 540
    YVS=YV(IYM1)
    YGS=YG(IYGS)
    YGN=YV(IY)+(YV(IY)-YG(IY))
    IF(IN.NE.2) YGN=YV(IY)
    GO TO 550
540 YVS=YV(IYVS)
    YGN=YG(IYGN)
    YGS=YG(IYGS)
550 CONTINUE
C
    IF(.NOT.TESTIN) GO TO 9100
    WRITE(12,9101) YVN,YVS,YGN,YGS
9101 FORMAT(1X,"YVN,YVS,YGN,YGS :",1X,4(1PE10.3,2X),/)
9100 CONTINUE
C
C    CHAPTER 6. - DETERMINING THE PARAMETERS REQUIRED.
C
    YS=(YP(IP)-YGS)/(YGN-YGS)
    YW=(YP(IP)-YVS)/(YVN-YVS)
    XS=(XP(IP)-XUW)/(XUE-XUW)
    XW=(XP(IP)-XGW)/(XGE-XGW)
    YN=1.0-YS
    YE=1.0-YW
    XN=1.0-XS
    XE=1.0-XW
    YSXS=YS*XS
    YSXN=YS*XN
    YNXS=YN*XS
    YNXN=YN*XN
    XWYW=XW*YW
    XWYE=XW*YE
    XEYW=XE*YW
    XEYE=XE*YE
C
C... DETERMINE THE VELOCITY COMPONENTS OF THE PARTICLE.
    UP=YSXS*UNE+YSXN*UNW+YNXS*USE+YNXN*USW
    VP=XWYW*VNE+XWYE*VSE+XEYW*VNW+XEYE*VSW
C
```

```

      IF(.NOT.TESTIN) GO TO 9110
      WRITE(12,9111) UP,VP
9111 FORMAT(1X,"UP, VP :",1X,2(1PE10.3,2X),/)
9110 CONTINUE
C
C      CHAPTER 7.- COMPUTE THE NEW POSITION OF THE PARTICLE.
C
      IF(.NOT.CARTES.AND.IPLANE.NE.2) GO TO 700
      XP(IP)=XP(IP)+UP*DELTAT
      GO TO 710
700 XP(IP)=XP(IP)+UP/(YP(IP)+RINNER)*DELTAT
710 YP(IP)=YP(IP)+VP*DELTAT
C
      IF(.NOT.TESTIN) GO TO 9120
      WRITE(12,9121) XP(IP),YP(IP)
9121 FORMAT(1X,"NEW XP AND YP :",1X,2(1PE10.3,2X),/)
9120 CONTINUE
C
C... CLOSE PARTICLES-DO-LOOP
1000 CONTINUE
      NPRIN=5
      IF(ISTP.GT.20) NPRIN=5
      IF(MOD(ISTP,NPRIN).NE.0) GO TO 777
      WRITE(22,998) ISTP
998 FORMAT(5X,"ISTP=",3X,I4)
C      IF(ISTP.NE.LSTEP) GO TO 777
      WRITE(22,999) (YP(IA),IA=1,NPMAX)
      WRITE(22,999) (XP(IA),IA=1,NPMAX)
999 FORMAT(1P10E12.3)
777 CONTINUE
C
C      CHAPTER 8.- PRINT STATEMENTS.
C
      NPRIN=5
      IF(MOD(ISTP,NPRIN).NE.0) GO TO 800
      WRITE(12,802) ISTP
802 FORMAT(5X,"*** OUTPUT FROM TRACK AT ISTP =",1X,I3,/)
      IF(NPMAX.GT.101) GO TO 888
      DO 800 IP=1,NPMAX
      IF(IPLANE.EQ.2) GO TO 805
      WRITE(12,801)IP,XP(IP),YP(IP)
801 FORMAT(5X,"PARTICLE",1X,I3,1X,"X-COORDINATE=",1X,1PE10.3,
12X,"Y-COORDINATE",1X,1PE10.3)
      GO TO 800
805 WRITE(12,806) IP,XP(IP),YP(IP)
800 CONTINUE
      GO TO 889
806 FORMAT(5X,"PARTICLE",1X,I3,1X,"Z-COORDINATE=",1X,1PE10.3,
12X,"Y-COORDINATE",1X,1PE10.3)
C 888 NPM30=NPMAX-30
C      IF(ISTP.EQ.100) GO TO 890
      DO 889 IP=NPM30,NPMAX
      WRITE(12,806) IP,XP(IP),YP(IP)
889 CONTINUE
C 890 WRITE(12,806) IP,XP(IP),YP(IP)
C
      NX=NXSTO
C

```

C CHAPTER 9. - PLOTTING STATEMENTS.

C

```
NPRIN=5
IF(MOD(ISTP,NPRIN).NE.0) GO TO 900
IF(.NOT.CARTES.AND.IPLANE.NE.2) RETURN
DO 901 IP=1,NPMAX
  XPLOT(IP)=XP(IP)
  YPLOT(IP,1)=YP(IP)
```

901 CONTINUE

C--- FOR PLOTTING WATER LEVAL

```
IWL=NPMAX+4
NPM4=IWL
NPM5=NPM4+19
XWAT=6.4E-2+7.683E-4*0.5*ISTP
DO 902 IP=NPM4,NPM5
  XPLOT(IP)=XWAT
  YPLOT(NPM4-1,1)=1.25E-3
  DY=(9.E-2-1.25E-3)/FLOAT(20)
```

902 YPLOT(IP,1)=YPLOT(IP-1,1)+DY

```
XAXIS=4HX-DI
YAXIS(1)=4HY-DI
IF(IPLANE.EQ.2) XAXIS=4HZ-DI
NC=1
```

```
NPM1=NPMAX+1
NPM2=NPMAX+2
NPM3=NPMAX+3
XPLOT(NPM1)=2.8E-2
YPLOT(NPM1,1)=1.25E-3
XPLOT(NPM2)=2.0E-1
```

C XPLOT(NPM2)=6.038E-2+7.658E-4*0.5*ISTP

```
YPLOT(NPM2,1)=1.25E-3
XPLOT(NPM3)=2.8E-2
YPLOT(NPM3,1)=9.E-2
```

CALL

PLOTSJ(ISTP,XPLOT,NPM5,NPM5,XAXIS,YPLOT,NC,NC,YAXIS)

900 CONTINUE

C

```
RETURN
END
```